P517368.PDF [Page: 4 of 68]

Copy No. 12

DEFENCE R&D DÉFENSE

VVES Model Generator and Graphical User Interface Application Version 1.0 User's Manual

Polak, M.F., Jiang, L.

Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada, B3J 3J8

Contract No W7707-008286/001/HAL

Defence R&D Canada

Contractor Report

DREA CR 2001-142

January 2002

National Defence Défense nationale

Canada

P517368.PDF [Page: 5 of 68]

Copy No: _____

VVES Model Generator and Graphical User Interface Application Version 1.0 User's Manual

Polak, M.F., Jiang, L.
Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia, Canada, B3J 3J8

Contract No. W7707-008286/001/HAL

Defence Research Establishment Atlantic

Contractor Report
DREA CR 2001-142
January 2002

P517368.PDF [Page: 6 of 68]

104-06260/

Authors

M.F. Polak L. Jiang

Contract Scientific Authority

D.C. Stredulinsky J.P Szabo

Approved by

J.L. Kennedy

Head, Warship Signatures & Safety

Approved for release by

Kirk Foster

Chair/DRP

The scientific or technical validity of this Contract Report is entirely the responsibility of the contractor and the contents do not necessarily have the approval or endorsement of Defence R&D Canada.

© Her Majesty the Queen as represented by the Minister of National Defence, 2002

© Sa majesté la reine, représentée par le ministre de la Défense nationale, 2002

P517368.PDF [Page: 7 of 68]

Abstract

The VVES (Vibration of Visco-elastic and Elastic Structures) suite of programs is a group of Matlab codes for analysis and design of engine isolation systems composed of engine, raft and ground which are connected by visco-elastic isolators. Among these programs, VVESANALYSIS allows the dynamic characteristics of isolators to be described by frequency-dependent complex material properties. The VVESDYNSTIFF and VVES4POLE require input of the directly measured dynamic stiffnesses and the 4 pole parameter matrices, respectively.

In order to simplify the construction of input files and provide an alternate method to view the VVES analysis results, a model generator and graphical user interface (GUI) for the VVES programs has been recently developed called VIMGEN (Vibration Isolation Model GENerator). VIMGEN was developed for the Windows environment using the DREA Hierarchical Object Oriented Developer's toolkit (HOOD). This user's manual provides instructions on using the VVES model generator and GUI. The reader of this report is assumed to have a reasonable familiarity with the VVES programs and the HOOD toolkit's core components.

Résumé

La famille de logiciels VVES (*Vibration of Visco-elastic and Elastic Structures* : vibrations des structures visco-élastiques et élastiques) regroupe des programmes MATLAB conçus pour analyser et concevoir des systèmes d'atténuation des vibrations des ensembles moteur-radier-coque solidarisés par des isolateurs visco-élastiques. Un de ces programmes, VVESANALYSIS, utilise les propriétés complexes des matériaux, lesquelles varient avec la fréquence, pour prédire les caractéristiques dynamiques des isolateurs. L'exécution de VVESDYNSTIFF requiert l'entrée des rigidités dynamiques mesurées directement et celle de VVES4POLE, des matrices quadripolaires de paramètres.

Afin de faciliter la création des fichiers d'entrée et fournir une autre méthode de visualisation des résultats de l'analyse réalisée par VVES, nous avons récemment élaboré, pour cette famille de programmes, le logiciel VIMGEN qui englobe un générateur de modèles et une interface graphique. (VIMGEN : *Vibration Isolation Model GENerator* ou générateur de modèles d'atténuation des vibrations). Nous avons élaboré ce programme, pour l'environnement Windows, à l'aide des outils HOOD (*Hierarchical Object Oriented Developer's toolkit* : la trousse de producteur de logiciels hiérarchiques orientés objet) du CRDA. Le présent guide contient des renseignements sur l'utilisation du générateur de modèle VVES et l'interface graphique. Le lecteur devra connaître raisonnablement bien les programmes VVES et les éléments centraux de la trousse logicielle HOOD.

P517368.PDF [Page: 8 of 68]

Executive summary

Background:

Marine diesel engines are supported by mounts that are designed to provide both structural rigidity and vibration isolation. Structural rigidity is required in order to maintain alignment of connecting shafts and piping; and vibration isolation is required in order to minimize the vibrations generated by the engine from being transmitted to the rest of the vessel and beyond. The vibration isolation materials currently in service on Canadian vessels, natural rubber and chloroprene rubber, experience large changes in their mechanical properties when exposed to lubricating oil or fuel. Replacing these materials with alternative elastomers could prevent alignment problems due to mount degradation, as well as reduce maintenance costs by increasing the length of time between mount changes. Replacement mounts must have a higher resistance to hydrocarbons and they must provide optimal vibration isolation. The optimal design of such mounts from a vibration viewpoint is complicated by the complex constitutive characteristics of the elastomers used in the construction of the mounts, the static compression of the mounts, and also by the complex nature of the three-dimensional vibration motion of the engine. DREA has developed sophisticated software for modeling engine vibration isolation systems. Although this code, called VVES (Vibration of Visco-elastic and Elastic Structures), provides a powerful tool for characterizing the dynamic properties of vibration isolation systems. It was not convenient to use VVES for practical engineering problems due to the large number of input files it requires.

Principal Results:

The principal result arising from this work is the development of a computer program called VIMGEN (Vibration Isolation Model GENerator), which is a GUI based model generator and control program for running the VVES software and visualization of results of analysis of engine/raft/foundation systems. This report is a user's manual for the VIMGEN software.

Significance of Results:

A sophisticated computational tool has been developed that will be of value in the optimal design of new isolators for marine diesel engines. The GUI based tools developed in this work will make model generation and running of the VVES code much easier and faster, especially for the novice or infrequent user, and should reduce errors in model generation.

Polak, M.F., Jiang, L. 2002. VVES Model Generator and Graphical User Interface Application Version 1.0 User's Manual. DREA CR 2001-142. Defence Research Establishment Atlantic.

P517368.PDF [Page: 9 of 68]

Sommaire

Contexte:

Les diesels marins reposent sur des supports dont la fonction est double : assurer la rigidité structurale et amortir les vibrations. La rigidité structurale permet de maintenir l'alignement des arbres et des conduites, tandis que l'amortissement des vibrations permet d'isoler le bâtiment et l'environnement des trépidations du moteur. Or, les propriétés mécaniques des matériaux antivibratiles couramment utilisés dans les navires canadiens — le caoutchouc naturel et le néoprène — sont fortement altérées par le contact avec l'huile lubrifiante ou le carburant. Substituer ces matériaux par d'autres élastomères pourrait prévenir les désalignements causés par la dégradation des supports et, en allongeant leur vie utile, réduire les coûts d'entretien. Les blocs de remplacement devront mieux résister aux hydrocarbures et assurer l'amortissement optimal des vibrations. Du point de vue de l'atténuation des vibrations, la conception optimale de ces supports est compliquée par les caractéristiques constitutives complexes des élastomères utilisés dans leur fabrication, la compression statique des supports et la nature complexe des mouvements vibratoires du moteur dans les trois dimensions. Le CRDA a produit un logiciel perfectionné, VVES (*Vibration of Visco-elastic and Elastic Structures* : vibrations des structures viscoélastiques et élastiques), pour modéliser les systèmes antivibrations des moteurs. Bien que ce logiciel soit un outil puissant de caractérisation des propriétés dynamiques des systèmes antivibrations, il n'était pas pratique de l'utiliser pour résoudre des problèmes concrets d'ingénierie à cause du nombre élevé de fichiers d'entrée qu'il exigeait.

Résultats principaux:

Le résultat principal de ce travail est la création de VIMGEN (*Vibration Isolation Model GENerator* : générateur de modèles d'atténuation des vibrations), un logiciel qui combine un générateur de modèles à interface graphique ainsi qu'un programme de commande permettant d'exécuter le logiciel VVES, et la visualisation des résultats de l'analyse de l'ensemble moteur-radier-coque. Le présent document est le guide de l'utilisateur du logiciel VIMGEN.

Importance des résultats:

Nous avons élaboré un outil informatique perfectionné qui sera utile pour la conception optimale de nouveaux supports antivibrations des diesels marins. Ces outils qui exploitent une interface graphique élaborée au cours de ce travail faciliteront et accéléreront l'exécution du logiciel VVES, en particulier pour les utilisateurs débutants ou occasionnels, et devraient réduire le nombre d'erreurs dans la production de modèles.

Polak, M.F. et Jiang, L. 2002. *VVES Model Generator and Graphical User Interface Application Version 1.0 User's Manual.* DREA CR 2001-142. Centre de recherches pour la Défense Atlantique.

P517368.PDF [Page: 10 of 68]

TABLE OF CONTENTS

TABLE OF CONTENTS v
LIST OF FIGURES vii
1. INTRODUCTION 1
2. CREATING A NEW PROJECT 1
3. OPENING AN EXISTING PROJECT 2
4. CHANGING PROJECT SETTINGS 2
5. INSERTING A MODEL 2
5.1 Defining the Engine 2
5.2 Selecting the Isolator Type 3
5.3 Defining the Isolators 3
5.3.1 Rectilinear Isolator Properties 3
5.3.2 Solid Cone Isolator Properties 3
5.3.3 Hollow Cone Isolator Properties 4
5.3.4 User-defined Isolator Properties 4
5.3.5 Isolator Description 4
5.3.6 Isolator Location 4
5.3.7 Isolator Material 4
5.3.7.1 Material database manager 4
5.3.7.2 Experimentally measured material properties 5
5.3.7.3 Havrilak-Nagami material model 5
5.3.7.4 Rational polynomial material model 5
5.3.8 Effect of Pre-stress 5
5.4 Choosing the Lower Component 5
5.5 Defining a Rigid Raft 6
5.6 Defining a Beam Raft 6
5.6.1 Beam Raft Plane 6
5.6.2 Lateral Beams 6
5.6.3 Longitudinal Beams 7
5.6.4 Beam Attachment Points 7
5.6.5 Beam Material and Geometry Type 7
5.7 Defining the Ground Plane 7
6. MODIFYING THE MODEL DIRECTLY 7
6.1 Engine Properties 7
6.2 Rigid Raft Properties 8
6.3 Beam Raft Properties 8
6.4 Common Isolator Properties 8
6.4.1 Isolator Reference 8
6.4.2 Isolator Attachment Points 8
6.4.3 Isolator Orientation 8
6.4.4 Isolator Color 9
6.4.5 Isolator Condensed Stiffness 9
6 4.6 Isolator Condensed Mass 9
6.4.7 Isolator Direct Complex Stiffness 9
6.4.8 Isolator 4-Pole Matrices 9

6.5 Generic Beam Properties 10
6.5.1 Generic Beam Material 10
6.5.2 Generic Beam Type 10
6.5.3 Generic Beam Attachments 10
6.5.4 Generic Beam Orientation 10
6.5.5 Generic Beam Drawing Mode 10
6.6 Lumped Mass Properties 10
6.7 Attachment Point Properties 11
6.8 Ground Point Properties 11
6.9 Ground Plane Properties 11
7. MODIFYING THE MODEL USING THE SCENE GRAPH 12
7.1 Show 12
7.2 Delete 12
7.3 Copy 12
7.4 Move: Isolator Layers 12
8. CONFIGURING MATLAB SETTINGS 12
9. CONFIGURING THE ANALYSIS 12
10. GENERATING INPUT FILES 13
11. RUNNING AN ANALYSIS 13
12. VIEWING RESULT PLOTS 13
13. VIEWING FREQUENCIES 14
14. VIEWING ANIMATION 14
15. VIEWING NODE NUMBERS 14
16. EDITTING DATABASES 14
17. INSTALLATION PROCEDURE 14
17.1 Some Known Issues Regarding Proper Installation 14
18. REFERENCES 15
APPENDIX A ..... A

P517368.PDF [Page: 12 of 68]

LIST OF FIGURES

Figure 1. New Project Dialog ..... A
Figure 2. Project Settings Dialog ..... A
Figure 3. Model Wizard: Engine Definition ..... B
Figure 4. Model Wizard: Engine Mount Points ..... B
Figure 5. Model Wizard: Isolator Type Selection C
Figure 6. Model Wizard: Rectilinear Isolator Geometry C
Figure 7. Model Wizard: Solid Cone Isolator Geometry ..... D
Figure 8. Model Wizard: Hollow Cone Isolator Geometry ..... D
Figure 9. Model Wizard: Isolator Definition – Description ..... E
Figure 10. Model Wizard: Isolator Definition – Location ..... E
Figure 11. Model Wizard: Isolator Definition – Material ..... F
Figure 12. Model Wizard: Isolator Definition – Pre-stress ..... F
Figure 13. Model Wizard: Attachment Choice ..... G
Figure 14. Model Wizard: Rigid Raft Definition ..... G
Figure 15. Model Wizard: Beam Raft – Plane Definition ..... H
Figure 16. Model Wizard: Beam Raft Definition – Lateral Beams ..... H
Figure 17. Model Wizard: Beam Raft Definition – Longitudinal Beams I
Figure 18. Model Wizard: Beam Raft Definition – Attachments I
Figure 19. Model Wizard: Beam Raft Definition – Material ..... J
Figure 20. Model Wizard: Ground Plane Definition ..... J
Figure 21. Frequency Dependent Material Database Manager ..... K
Figure 22. Experimental Viscoelastic Material Dialog ..... K
Figure 23. Havrilak Negami Viscoelastic Material Dialog L
Figure 24. Rational Polynomial Viscoelastic Material Dialog L
Figure 25. G1 and G2 Dialog ..... M
Figure 26. Frequency Range Dialog ..... M
Figure 27. Young's Modulus and Poisson's Ratio Dialog ..... N
Figure 28. Engine Properties Dialog ..... N
Figure 29. Attachment Points Dialog ..... O
Figure 30. Rigid Raft Properties Dialog ..... O
Figure 31. Beam Raft Properties Dialog ..... P
Figure 32. Isolator Reference Dialog ..... P
Figure 33. Isolator Attachments Dialog ..... Q
Figure 34. Isolator Orientation Dialog ..... Q
Figure 35. Isolator Color Dialog ..... R
Figure 36. Isolator Condensed Stiffness Dialog ..... R
Figure 37. Isolator Condensed Mass Dialog ..... S
Figure 38. Isolator Complex Stiffness Dialog ..... S
Figure 39. Isolator 4-pole Matrices Dialog ..... T
Figure 40. Isolator Material Dialog (DYNSTFF,4-POLE) ..... T
Figure 41. Generic Beam Material Dialog ..... U
Figure 42. Generic Beam Type Dialog ..... U
Figure 43. Generic Beam Attachments Dialog V
Figure 44. Generic Beam Orientation Dialog V

P517368.PDF [Page: 13 of 68]

Figure 45. Generic Beam Drawing Mode Dialog ..... W
Figure 46. Lumped Mass Properties Dialog ..... W
Figure 47. Attachment Point Properties Dialog X
Figure 48. Ground Point Properties Dialog X
Figure 49. Ground Plane Properties Dialog ..... Y
Figure 50. Attachment Point Picking Dialog ..... Y
Figure 51. Color Modification Dialog ..... Z
Figure 52. Scene Graph Tree View ..... AA
Figure 53. Scene Graph Tree Popup Menu ..... AA
Figure 54. Matlab Configuration Dialog ..... BB
Figure 55. Analysis Options Dialog ..... BB
Figure 56. System Matrices Settings CC
Figure 57. Free Vibration Settings CC
Figure 58. Frequency Response Functions Settings ..... DD
Figure 59. Force Transmissibility Settings ..... DD
Figure 60. Displacement Response Settings ..... EE
Figure 61. Quasi-static Motion Settings ..... EE
Figure 62. Calculation Dialog ..... FF
Figure 63. Adding Plots ..... FF
Figure 64. Selecting a Chart for Display ..... GG
Figure 65. Sample Chart ..... GG
Figure 66. Sample Chart Data ..... HH
Figure 67. Animation Dialog ..... HH
Figure 68. Animation in Progress II
Figure 69. The File Menu II
Figure 70. The View Menu ..... JJ
Figure 71. The Insert Menu ..... JJ
Figure 72. The Analysis Menu ..... JJ
Figure 73. The Results Menu ..... KK
Figure 74. The Project Menu ..... KK
Figure 75. The Tools Menu ..... KK

P517368.PDF [Page: 14 of 68]

1. INTRODUCTION

The VVES (Vibration of Visco-elastic and Elastic Structures) suite of programs [1] contains a group of Matlab computer codes for analysis of the vibration and transmissibility characteristics of a marine engine supported by visco-elastic isolators when subjected to harmonic excitation. In the basic structural configuration considered in these programs, an engine block, which is represented by a rigid body, is supported on a foundation raft by visco-elastic isolators, and the raft itself is connected to ground by a second layer of isolators. The raft can be either rigid or flexible, and the isolators are composed of visco-elastic materials, whose properties can be characterised by complex Young's modulus and Poisson's ratio for harmonic responses.

The VVES suite includes three programs, namely VVESANALYSIS, VVESDYNSTIFF and VVES4POLE. The difference between them is the method used for describing the complex frequency-dependent material properties of the visco-elastic isolators. In the first program, VVESANALYSIS, complex frequency-dependent material constants, such as Young's modulus and Poisson's ratio or shear and bulk moduli, are defined through data tables or analytical formulas, whereas in the last two programs, experimentally measured dynamic stiffness matrices or 4 pole parameter matrices must be provided.

In order to simplify the process of using these VVES programs, a special purpose model generator and graphical user interface had been developed based on the HOOD toolkit [2]. In this document, detailed instructions are given on how to use this VVES model generator to create input data, run VVES programs and display analysis results. It should be noted that in this report, only those HOOD functionality which are directly related to the present application are described.

To enhance the usability of this manual, all the dialog boxes provided in the presently developed VVES model generator are screen-captured and presented in the Appendix. These figures are arranged in the order that these boxes are to be accessed by a typical user. This order is as follows:

Figure 1 – 2:	Project setting
Figure 3 – 20:	Model wizard
Figure 21 – 27:	Frequency-dependent material definition
Figure 28 – 51:	Additional properties of structural components
Figure 52 - 53:	Scene graph tree
Figure 54 – 61:	VVES analysis control parameters
Figure 62 – 68:	Post-processing
Figure 69 – 75:	Toolbar and menus

2. CREATING A NEW PROJECT

The first step in creating a VVES model is to create a new project. Select the New item from the File menu (Figure 69) or click the New button on the toolbar. Both methods display the New Project dialog (Figure 1).

P517368.PDF [Page: 15 of 68]

Select a directory for the new project by clicking on the button at the end of the Project Directory field. The chosen directory should be initially empty as a large number of data files could potentially be accumulated there.

Next, enter a project name. The project name cannot contain spaces. The description and author fields are optional and so do not have to contain any text.

The last field determines the type of simulation. The options available are the following: analysis, dynamic stiffness, and 4-pole. The choice made now cannot be changed later. To create a project for the same engine isolation system for being analysed by another VVES program, use the Export option from the File menu.

3. OPENING AN EXISTING PROJECT

To continue working with an existing model, load the file by either selecting the Open item from the File menu (Figure 69) or clicking the Open button on the toolbar.

4. CHANGING PROJECT SETTINGS

To change the project description or author or to see the current project settings, select the Settings item from the Project menu (Figure 74). The Project Settings dialog will be displayed (Figure 2).

5. INSERTING A MODEL

The simplest method of creating a new model is to use the model wizard. To insert a new model select the Model item from the Insert menu (Figure 71). The model wizard will display a sequence of pages that will gather information required for constructing a model.

5.1 Defining the Engine

The first page (Figure 3) shown by the model wizard is the engine definition page. The origin of the local co-ordinate system is attached to the mid point of the back bottom edge of the engine block such that the width is divided symmetrically by the x'-axis. The local x' and y'-axis are defined along the length and width of the block, respectively. The local z'-axis is along the vertical direction and points upward. In the present version of the VVES model generator, it is assumed that the global co-ordinate system initially coincide with the local system of the engine. However, the location of the engine can be modified later if required.

In the VVES programs, the engine is modelled as a rigid body. When the dialog box is displayed, default values have been used for all the physical properties required. If the mass or a dimension is altered the values for the center of mass and mass moments of inertia should be recalculated by clicking the 'Calc' button. All calculated values may be changed manually.

The attachments points (mount points) are specified next (Figure 4). The wizard assumes a symmetric spacing of points but these locations can be adjusted after the points have been created.

P517368.PDF [Page: 16 of 68]

5.2 Selecting the Isolator Type

Below the engine is a layer of isolators. Figure 5 shows the four types of isolators that may be used, including rectilinear, hollow cone, solid cone, and user defined isolators. The following two pages will define one isolator which will be used as a template for all the isolators in this layer. Additional information will be required to fully specify the isolators but this will have to be specified after the model has been created.

With the exception of the VVESANALYSIS code, all the isolators in the same layer must be of the same material and shape. The option to keep all of the isolators identical is also offered as a convenience for the VVESANALYSIS program.

5.3 Defining the Isolators

The first page of the isolator definition is the geometry page. Figures 6, 7, and 8 are the geometry pages for rectilinear, hollow cone, and solid cone isolators, which are referred as Type I isolators in the VVES programs. There is no geometry information (and thus no page) required for the isolators with user defined condensed stiffness and mass matrices. These latter isolators are referred to as Type II isolators in the VVES codes.

For display purposes, the geometric shape of the isolator may be used or the isolator can simply be represented as a straight line connecting its two attachment points. By default, the isolator shape is drawn.

The VVES codes can provide an approximation for the condensed stiffness and mass matrices of an isolator by conducting a linear static finite element analysis. This method of calculating the condensed matrices is used by default. If the user wants to enter the condensed stiffness and mass matrices, it can be done after the model has been created.

5.3.1 Rectilinear Isolator Properties

Figure 6 shows the definition page for the rectilinear isolator. The dimensions of the isolator are defined and used to draw the shape in the model view. The isolator can also be represented as a simple line connecting the isolator's attachment points by clearing the 'draw shape' check box. The finite element mesh density required for the computation of the condensed stiffness matrix is also specified on this page.

5.3.2 Solid Cone Isolator Properties

Figure 7 shows the definition page for the solid cone isolator. The dimensions of the isolator are defined and used to draw the shape in the model view. The isolator can also be represented as a simple line connecting the isolator's attachment points by clearing the 'draw shape' check box. The finite element mesh density required for the computation of the condensed stiffness matrix is also specified on this page.

P517368.PDF [Page: 17 of 68]

5.3.3 Hollow Cone Isolator Properties

Figure 8 shows the definition page for the hollow cone isolator. The dimensions of the isolator are defined and used to draw the shape in the model view. The isolator can also be represented as simple line connecting the isolator's attachment points by clearing the 'draw shape' check box. The finite element mesh density required for the computation of the condensed stiffness matrix is also specified on this page.

5.3.4 User-defined Isolator Properties

There are no additional properties for these Type II isolators.

5.3.5 Isolator Description

Each isolator can have a unique description, which can contain multiple lines of text (Figure 9).

5.3.6 Isolator Location

The isolator location refers to the offset of the bottom of the visco-elastic component of the isolator from the base attachment point (Figure 10). An offset is used to represent an additional layer of material (such as a steel base plate) that is not explicitly modelled.

5.3.7 Isolator Material

Each isolator is constructed from a visco-elastic material, which has complex frequency-dependent material properties. For the VVESANALYSIS program, these frequency-dependent material properties must be provided in either the form of Young's modulus and Poisson's ratio or equivalently, the form of shear and bulk moduli. The materials available are listed in the combo box (Figure 11) with the currently selected material being displayed. Additional materials may be added by clicking the 'New/Edit' button to display the material database manager (Figure 21).

For the VVESDYNSTIFF and VVES4POLE programs, the visco-elastic material properties of the isolators are characterised by the frequency-dependent direct complex stiffness and 4-pole parameter matrices, respectively. The dialog boxes for defining these material data are presented in Sections 6.4.7 and 6.4.8.

5.3.7.1 Material database manager

The material database manager is used to specify and modify material types. It provides a list of all the material properties which are currently available in the database. The three types of material data can be considered, including: Experimentally measured data (Figure 22), the Havrilak-Nagami model (Figure 23), and the Rational Polynomial model (Figure 24). Through the material database manager dialog box (Figure 21), the user can view, edit, insert and delete material properties. The filter, which is shown in the left side of the dialog box, is a standard HOOD capability for imposing criteria when searching through the database. This feature is not

P517368.PDF [Page: 18 of 68]

used in the current version of the VVES model generator.

5.3.7.2 Experimentally measured material properties

The experimentally measured material properties are defined by complex Young's modulus and Poisson's ratio at a series of loading frequencies. The table can be directly specified or that data can be loaded from a text file. Data from the table can also be saved as a text file. Clicking on the Open and Save buttons at the bottom left of the page causes the pop up of the standard window's browser, which allows the user to define filename and path. The format of the file is tab delimited entries corresponding to the order of the columns in the table, namely frequency, real part of Young's modulus, imaginary part of Young's modulus, real part of Poisson's ratio and imaginary part of Poisson's ratio. In Figure 22, the third image at the bottom left of the dialog correspond to deleting a row of data.

The material data in terms of Young's modulus and Poisson's ratio can be viewed in the shear modulus G1 and bulk modulus G2 format by pressing the 'View G1, G2' button. This will cause the display of another table (Figure 25). Data in this format can also be saved by pressing the save file button.

5.3.7.3 Havrilak-Nagami material model

Figure 23 shows the definition dialog for material properties described by the Havrilak-Nagami model. To view the corresponding Young's modulus and Poisson's ratio for this material, press the 'View E,v' button. The frequency range dialog shown in Figure 26 will be displayed. Specify the range of frequency to be calculated and the number of points to be generated and then press the 'OK' button, a table with the calculated data will be displayed (Figure 27). This data can be saved to a file.

5.3.7.4 Rational polynomial material model

Figure 24 shows the definition dialog for the Rational Polynomial material model. To view the corresponding Young's modulus and Poisson's ratio for this material press the 'View E,v' button. The frequency range dialog shown in Figure 26 will be displayed. Specify the range of frequency to be calculated and the number of points to be generated and then press the 'OK' button, a table with the calculated data will be displayed (Figure 27). This data can be saved to a file.

5.3.8 Effect of Pre-stress

Figure 12 shows the pre-stress definition dialog. If the effect of pre-stress is to be included, enter values for the fields shown. For type I isolators, the shape factor is calculated automatically from the isolator geometry.

5.4 Choosing the Lower Component

As the model is being constructed objects are being placed beneath the previously specified isolator layer. After each isolator layer has been defined a decision must be made to add a raft or

P517368.PDF [Page: 19 of 68]

connect the isolators to the ground. Figure 13 shows these available options.

5.5 Defining a Rigid Raft

The model wizard displayed the page shown in Figure 14 when a rigid raft is to be defined. The distance between the bottom surface of the previous structure (engine or another raft) and the top surface of the present raft is defined in this page as well as the raft's other physical properties. The origin of the local co-ordinate system is attached to the middle of the back bottom edge, in the same way as for the engine block, and the axes of the local system are assumed to be along the axes of the global co-ordinate system. When a rigid raft is initially generated, its centre of mass is aligned with the centre of mass of the engine block above it. But, this location can be changed if necessary.

The physical properties required for defining a rigid raft include dimensions, mass and mass moments of inertia. The length, width and height are in the directions of the local x', y' and z' axes. The default values of these dimensions are derived from the engine geometry. If the mass or a dimension is altered, the values for the centre of mass and mass moment of inertia should be recalculated by clicking the 'Calc' button. All calculated values may be changed manually.

The upper attachment points of the raft are defined based on the attachment points defined on the bottom surface of the previous structural component. The lower attachment points are specified in a manner identical to the engine mount points (Figure 4).

5.6 Defining a Beam Raft

A beam raft is a flexible raft composed of a number of lateral and longitudinal beams.

5.6.1 Beam Raft Plane

The mid-plane of the raft is described by the distance from the bottom surface of the previous structural component (engine or another raft) to the mid-plane of the present draft, and the dimensions of the bounding plane (Figure 15). The origin of the local co-ordinate system is defined at the mid point of the back edge of the bounding plane and the directions of the local axes are along those of the global system. When a beam raft was initially created, the centre of the bounding plane is aligned with the centre of mass of the engine. The centre lines of all beam elements contained in a beam raft are assumed to be within this mid-plane. The actual height of the raft is determined by the height of the beams themselves.

5.6.2 Lateral Beams

A beam type and a location along the local x'-axis define a lateral beam. Beam information may be entered explicitly into the table. If all the beams are of the same type and equally spaced, they may be added by selecting the desired type and number, and then clicking the 'Add' button. See Figure 16 for details.

5.6.3 Longitudinal Beams

A beam type and a location along the local y'-axis define a longitudinal beam. Beam information may be entered explicitly into the table. If all the beams are of the same type and equally spaced, they may be added by selecting the desired type and number, and then clicking the 'Add' button. See Figure 17 for details.

5.6.4 Beam Attachment Points

As shown in Figure 18, attachment points are initially generated at every intersection of lateral and longitudinal beams as well as at every point beneath the attachment points on the previous structure. If the distance between an attachment point and an intersection point is identified to be within the beam width, the intersection point is used and the other point is removed. Points may be explicitly added to the table or can be generated by selecting a beam and specifying the number of points to be added. The image in the right corner shows a representation of the raft with the actual number of lateral and longitudinal beams as well as the locations of the attachment points.

5.6.5 Beam Material and Geometry Type

All beam elements in a beam raft are assumed to be composed of the same material. The materials available are listed in the combo box (Figure 19). Materials may be specified or modified by pressing the 'New' button to display the material database manager.

The beam types available are shown in the list. To modify the available types, press the 'New' button next to the list. The beam database manager will then be displayed.

5.7 Defining the Ground Plane

Figure 20 shows the ground plane definition. The dimensions of the object above it (engine or raft) define the default size of the ground plane. Ground points will be automatically generated based on the attachment points of the engine or the lower attachment points of the last raft.

6. MODIFYING THE MODEL DIRECTLY

Components may be modified by double-clicking the component in the three-dimensional model view window. Alternately, the property page for the component may be shown by right-clicking on the node in the scene graph window as will be explained in Section 7.

6.1 Engine Properties

The engine property page is shown in Figure 28. Pressing the 'Define/Modify Attachment Points' button can modify the attachment points. The attachment points dialog is shown in Figure 29. The location of the points can be entered directly into the table or automatically generated. Specifying the information in the area at the top and pressing the 'Add' button can generate equally spaced points.

P517368.PDF [Page: 21 of 68]

The color of the engine can be changed by pressing the colored button to display the color modification dialog (Figure 51). Presently, only the Diffuse Color is used.

6.2 Rigid Raft Properties

The rigid raft property page is shown in Figure 30. Pressing the button corresponding to the upper or lower attachment points can modify the attachment points. The attachment points dialog is shown in Figure 29. The location of the points can be entered directly into the table or automatically generated. Specifying the information in the area at the top and pressing the 'Add' button can generate equally spaced points.

The color of the raft can be changed by pressing the colored button to display the color modification dialog (Figure 51). Presently, only the Diffuse Color is used.

6.3 Beam Raft Properties

The beam raft property page is shown in Figure 31. All of the beam properties are defined in the model wizard section 5.6.

6.4 Common Isolator Properties

The model wizard section 5.3 described a few of the isolator properties. The remaining properties are described in this Section.

6.4.1 Isolator Reference

An isolator may be quickly defined by setting the reference isolator from the isolator database (Figure 32). All of the isolator properties defined in the database isolator will be copied to the new isolator.

6.4.2 Isolator Attachment Points

An isolator must be attached to two points referred to as its upper and lower attachment points. Figure 33 shows the presently defined attachment points for an isolator. The two buttons next to each attachment point are used to select the attachment point. The first allows for selection of a point from a list (Figure 50). The second allows for direct picking of the point from the model view. NOTE: to select an attachment point using the direct picking method, hold down the right 'Ctrl' key and double-click the attachment point in the model view window.

6.4.3 Isolator Orientation

Figure 34 shows the definition page for the isolator orientation. The orientation of an isolator is defined by the local y'-z' plane formed by its upper and lower attachment points and a third point that is not co-linear with the attachment points. The local z'-direction is always along the line connecting the upper and lower attachment points and the third point can be defined in a variety of ways. If an existing point in the model is suitable for being used as the third point, it can be

selected by either specifying the point or picking it from the model view. NOTE: to select an attachment point using the direct picking method, hold down the right 'Ctrl' key and double-click the attachment point in the model view window.

An alternative method for defining the orientation plane is to use an orientation angle referenced to a base orientation plane which is defined by the local z'-axis and one of the global axes which are not parallel to the local z' direction.

6.4.4 Isolator Color

The page for defining the color of the isolators is shown in Figure 35. Pressing this colored button displays the color modification dialog (Figure 51). All isolators share the same color, so modifying one isolator's color will change the color of all isolators.

6.4.5 Isolator Condensed Stiffness

The condensed stiffness data page is shown in Figure 36. This information is only used for Type II isolators (for definition of isolator types, see Section 5.3). Only the components along the diagonal are used (and stored). Data may be read from a file or saved to a file. The format of the text file is the value of the six elements each separated by a tab.

6.4.6 Isolator Condensed Mass

The condensed mass data page is shown in Figure 37. This information is only used for Type II isolators. Only the components along the diagonal are used (and stored). Data may be read from a file or saved to a file. The format of the text file is the value of the six elements each separated by a tab.

6.4.7 Isolator Direct Complex Stiffness

The complex stiffness data are required for running the VVESDYNSTIFF program. The page for specifying these data is shown in Figure 38. Item 'Complex Stiffness' is displayed in the list at the left of the page if the dynamic stiffness analysis option was selected for the project. Data may be read from a file or saved to a file. The format of the text file is shown below:

Frequency K11real K11imag K22real K22imag K33real K33imag

One line of data of the format should be provided for each loading frequency.

6.4.8 Isolator 4-Pole Matrices

The 4-pole matrices data page is shown in Figure 39. Item '4-pole Matrices' is displayed in the list at the left of the page if the 4-pole analysis option was selected for the project. Data may be read from a file or saved to a file. The format of the text file is shown below (where 'r' refers to the real part of the value and 'i' to the imaginary part):

P517368.PDF [Page: 23 of 68]

Frequency X11r X11i X12r X12i X21r X21i X22r X22i Y11r Y11i Y12r Y12i Y21r Y21i Y22r Y22i Z11r Z11I Z12r Z12i Z21r Z21i Z22r Z22I

One line of data of the format should be provided for each loading frequency.

For both dynamic stiffness and 4-pole options, additional material properties are required, including mass density and Poisson's ratio at zero frequency. These material parameters can be input as shown in Figure 40.

6.5 Generic Beam Properties

Beam elements belonging to a beam raft were described previously in Section 5.6. The following sub-sections describe properties of beam elements that are not part of a beam raft. A generic beam can be inserted into a model by selecting 'Beam' from the Insert menu (Figure 71).

6.5.1 Generic Beam Material

Each beam has its own material property. The material is selected from the material database as shown in Figure 41. New material types can be added by activating the material database manager by using the 'New/Edit' button.

6.5.2 Generic Beam Type

As with the beams in a beam raft, beam cross-section properties are defined using beam types. The beam type for the current element is selected from the beam database as shown in Figure 42. New beam types can be added by activating the beam database manager using the 'New/Edit' button.

6.5.3 Generic Beam Attachments

Figure 43 shows the beam attachment points property page. Every individual beam element needs to be connected to a pair of attachment points, just like isolators.

6.5.4 Generic Beam Orientation

Figure 44 shows the beam orientation property page. The options for defining beam orientation are exactly the same as those for defining isolator orientation described before in Section 6.4.3.

6.5.5 Generic Beam Drawing Mode

The beam drawing options are shown in Figure 45. Each beam can be viewed as a straight line, a straight line with a two-dimensional cross-section plotted at the mid point, or as full three-dimensional beam. (Note: the orientation axis has not been implemented for this version)

6.6 Lumped Mass Properties

Figure 46 shows the definition page for a lumped mass. The attachment point can be selected

P517368.PDF [Page: 24 of 68]

from a list or by directly picking the attachment point from the model view. NOTE: to select an attachment point using the direct picking method, hold down the right 'Ctrl' key and double-click the attachment point in the model view window.

The mass moment of inertia can be either directly entered or calculated in two different ways, from a representative block or from the user-defined radii of gyration.

The shape of the lumped mass will be a box if both mass and mass moment of inertia have been specified. The size of the box will be representative of the properties, based on the assumption of uniform mass distribution. If the lumped mass has no moments of inertia defined, it will be represented as a sphere with a size determined by the user specified mass and the density of concrete.

The color of the lumped mass can be changed by pressing the colored button to display the color modification dialog (Figure 51).

6.7 Attachment Point Properties

Figure 47 shows the attachment point property page. An attachment point is represented as a sphere. Attachment points usually belong to engine, raft, or ground plane, but can also be directly inserted into the model by selecting 'Attachment Point' from the Insert menu (Figure 71).

The color of the attachment points can be changed by pressing the colored button to display the color modification dialog (Figure 51). Changing the color of any one attachment point changes the color for all attachment points.

6.8 Ground Point Properties

Figure 48 shows the ground point property page. A ground point is an attachment point with constraints. A description can also be entered for a ground point.

The color of the ground points can be changed by pressing the colored button on the attachment point page to display the color modification dialog (Figure 51). Since a ground point is a special type of attachment point, changing the color of any ground point changes the color for all attachment points as well.

6.9 Ground Plane Properties

Figure 49 shows the ground plane property page. The ground plane has planar dimensions and contains a collection of ground points. The ground points can be modified by pressing the 'Define/Modify Attachment Points' button.

The color of the ground plane can be changed by pressing the colored button to display the color modification dialog (Figure 51).

P517368.PDF [Page: 25 of 68]

7. MODIFYING THE MODEL USING THE SCENE GRAPH

The model view consists of a collection of objects stored in a hierarchy. This hierarchy is called the scene graph. A typical scene graph is shown in Figure 52. The scene graph is useful for performing certain operations on the model. Some of these operations can only be done through the scene graph while others can also be performed directly from the model view. Figure 53 shows the full list of operations that are available and is accessed by right-clicking on an entry in the list.

7.1 Show

This operation is identical to double clicking on an object in the model view with the left mouse button. The property page for the object is displayed.

7.2 Delete

Removes the object from the scene. Deleting an attachment point will cause connected isolators to be become partially undefined. Such isolators will be drawn as a sphere and will need to be reconnected to another attachment point before the model can be calculated.

7.3 Copy

Copying an object (such as an isolator) is an easy way to avoid having to re-specify all of the object's properties.

7.4 Move: Isolator Layers

Isolators in each layer are represented as children of the isolator layer parent. Changing the location of the isolator may violate this relationship and cause a problem with the subsequent calculation of the model. For instance, an isolator in a layer that is moved from its position as a child node to that of a sibling node will no longer have its properties synchronized with the other isolators that are still part of the isolator layer. This may cause a problem if the VVES code is expecting all isolators in the same layer to have consistent properties, such as VVESDYNSTIFF and VVES4POLE. As a result, only change the location of an isolator in the scene graph tree if you are sure the resulting relationship will be allowable for the given simulation.

8. CONFIGURING MATLAB SETTINGS

In order to run Matlab, the program must know where to find the VVES codes. Select 'Matlab Settings' from the Tool menu (Figure 75) to display the configuration dialog (Figure 54). Click on the buttons to select the folders containing the required information.

9. CONFIGURING THE ANALYSIS

The VVES codes have a number of configuration options depending on the type of results that should be calculated. Select 'Options' from the Analysis menu (Figure 72) to display the analysis

P517368.PDF [Page: 26 of 68]

options dialog (Figure 55).

Depending on the VVES program specified in the project, the choices may be restricted (few results can be calculated for the 4-pole model for instance). Check the box next to each result to be calculated. Click the 'Settings' button to specify information for each type of results. Each result has its own page of parameters shown in Figures 56 to 61. The meaning of the parameters can be found in Reference [1].

10. GENERATING INPUT FILES

When the model is fully specified, select 'Generate Input Files' from the Analysis menu (Figure 72) to create the VVES input files. The files will be created in the current project's directory.

11. RUNNING AN ANALYSIS

After generating the required input files, select 'Run Analysis' from the Analysis menu (Figure 72) to display the calculation dialog (Figure 62). Click the 'Start' button to begin the process. Matlab will be launched and the input files will be processed. The calculation may take a while depending on the speed of your machine and the complexity of the model. Upon completion of an analysis, a message will be displayed. Click 'Close' to return to the model.

12. VIEWING RESULT PLOTS

After running the analysis, a number of result files will have been produced. The first step in examining these results is to select which output files to use. Select 'Add Plot' from the Results menu (Figure 73) to display the dialog shown in Figure 63. The type of analysis files to search for is determined by the selection in the combo box. By default, this is set to be the same as the current project. Click on the button to display the directory selection dialog and change to the folder containing the results to be viewed. Press 'OK' at the correct folder. Results can be shown for any project that has been calculated not just the current project. The force direction and locations may not be valid for other projects if the geometry of the model is not the same as the current model. The check boxes will be enabled if the corresponding output files are available. Check all the results to be made available for plotting.

The results that were checked are now explicitly listed in the plot viewing selection dialog (Figure 64). This dialog can be returned to later by selecting 'View Plot' from the Results menu. Select the item in the list to be plotted and press 'Show Chart'. See Figure 65 for a sample chart.

The chart shows the magnitude and phase for one degree of freedom at a time. Select an item from the combo box at the top of the dialog to change the set of data being shown. Double-click on the curves to display the data at that point. All of the data for the chart can be seen in a tabular format by selecting the Data tab. Figure 66 shows an example of the chart data.

For force related data, an arrow indicating the direction of the force will be shown in the model view. Also, the response location will be shown as a sphere if the information is available.

P517368.PDF [Page: 27 of 68]

13. VIEWING FREQUENCIES

After a free vibration analysis calculation has been performed, the resulting frequencies can be viewed in a table. To see the table, select 'View Frequencies' from the Results menu (Figure 73).

14. VIEWING ANIMATION

An animation of the vibration analysis results can be seen by selecting View Animation from the Results menu (Figure 73). Before animation can occur, the mode must be selected. The animation dialog (Figure 67) provides a selection field for choosing the mode to animate. The number of modes (eigenvalues) can be configured in the analysis settings for the Free Vibration calculation. Press Animate to start the animation calculations. Once the calculations for the model have been made an animation window will appear (Figure 68). The animation will be displayed for about 15 seconds before the window closes. Pressing Animate again will display the animation window without repeating the calculation process.

15. VIEWING NODE NUMBERS

As an aide to the common task of node selection the node numbers for all attachment points in the model can be displayed. Select the Show Node Labels from the View menu (Figure 70) to enable or disable this feature.

16. EDITTING DATABASES

The material, isolator, frequency material, and beam databases can be accessed directly by selecting Edit Database from the Tools menu (Figure 75). You must have previously defined the location of each of the databases in order for the databases to be available for modification.

17. INSTALLATION PROCEDURE

1. Run the installation program and choose the installation directory.
2. Run VIMGEN.
3. Select the material, beam, frequency material, and isolator databases. Sample database files are provided with the installation and will be stored in sub-directories of the program installation directory. You may copy these files and store them elsewhere on your computer or reference existing database files if suitable files exist. (You will be prompted to specify database files the first time you create a model.)
4. Specify the location of the VVES script files. Use the Tools/Matlab Configuration menu option to configure the location of the script files. Once these settings are made they will be used for all subsequent Matlab simulations.

17.1 Some Known Issues Regarding Proper Installation

1. Matlab must be configured as a server on your computer in order for VIMGEN to use Matlab to run the VVES scripts. To register Matlab as a server run the following command from a command prompt:

P517368.PDF [Page: 28 of 68]

C:\> matlab /regserver

2. Win98 may exhibit a synchronization problem when VIMGEN attempts to launch Matlab to either run a VVES simulation or show a mode animation. Message boxes will appear with a message similar to: "Cannot connect to server" or "Server busy". Launching Matlab before using VIMGEN seems to resolve this problem in some cases. The exact nature of this problem has yet to be discovered.

18. REFERENCES

[1] S. G. HUTTON, "Matlab Computer Programs for Vibration and Control of Vibrations Transmitted from Marine Engines' Users Manual", Vibe-Tek Research Ltd., Vancouver, BC, Canada, March 2000.

[2] M. LICHODZIJEWSKI and T. MACADOM, "HOOD User's Manual", Martec Technical Report TR-99-05, February 1999.

P517368.PDF [Page: 29 of 68]

APPENDIX A

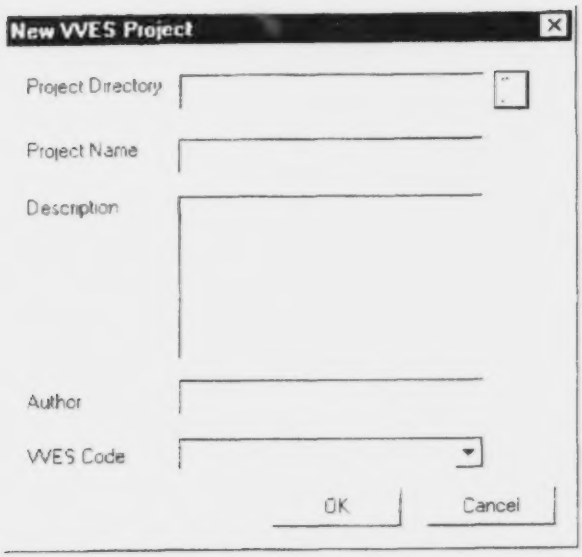

Figure 1. New Project Dialog

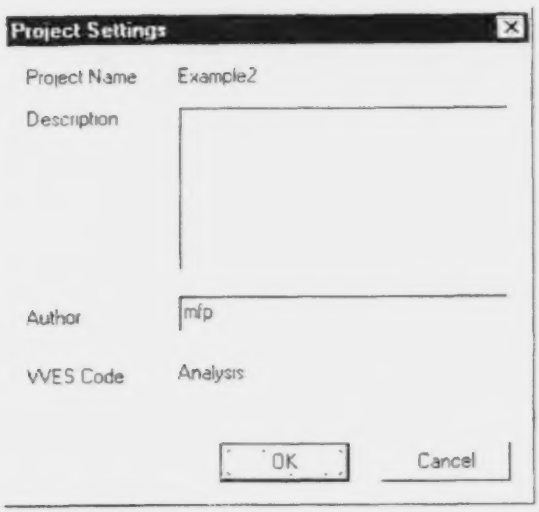

Figure 2. Project Settings Dialog

P517368.PDF [Page: 30 of 68]

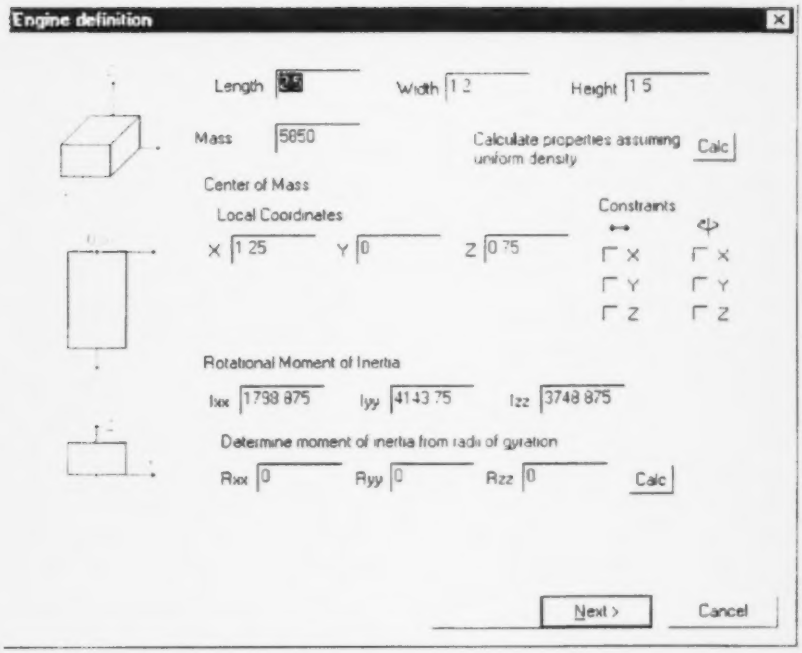

Figure 3. Model Wizard: Engine Definition

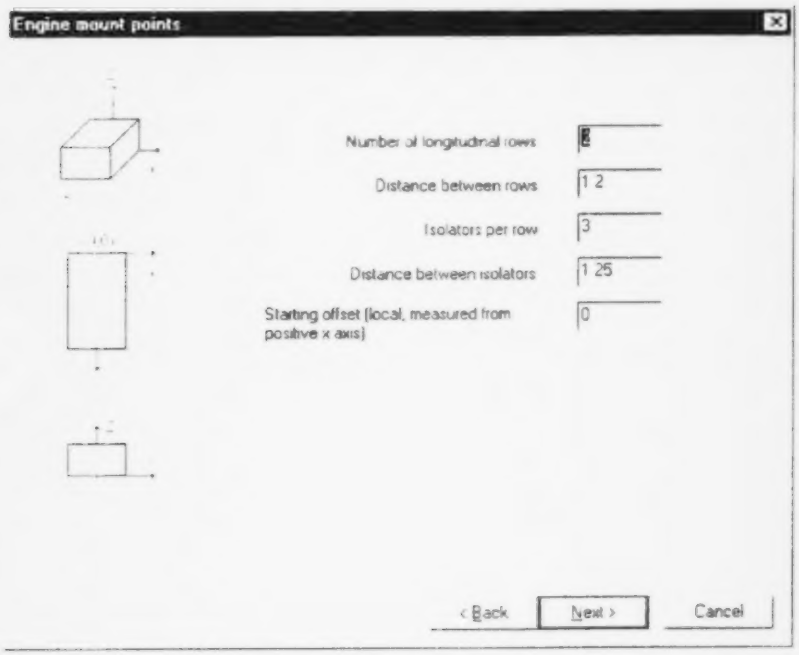
Engine mount points
Number of longitudinal rows 2
Distance between rows 1.2
Isolators per row 3
Distance between isolators 1.25
Starting offset (local, measured from positive x axis) 0
< Back
Next >
Cancel

Figure 4. Model Wizard: Engine Mount Points

P517368.PDF [Page: 31 of 68]

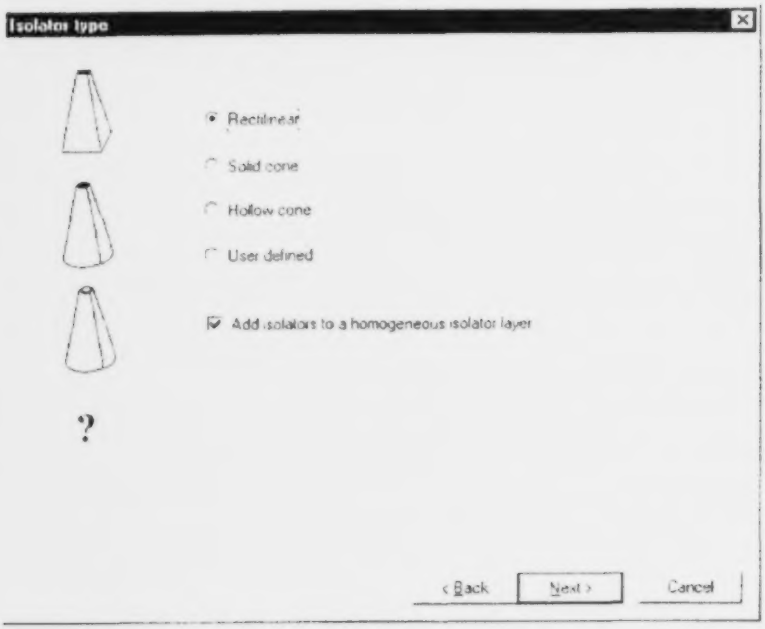

Figure 5. Model Wizard: Isolator Type Selection

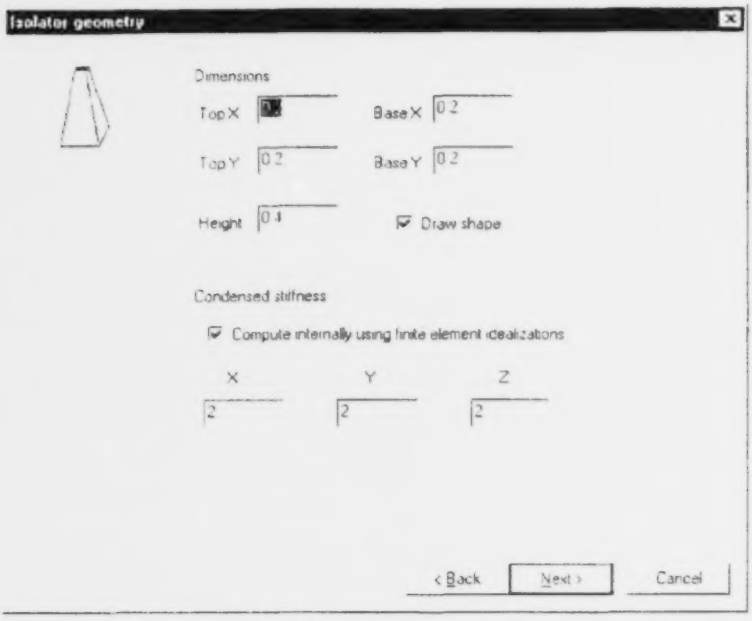

Figure 6. Model Wizard: Rectilinear Isolator Geometry

P517368.PDF [Page: 32 of 68]

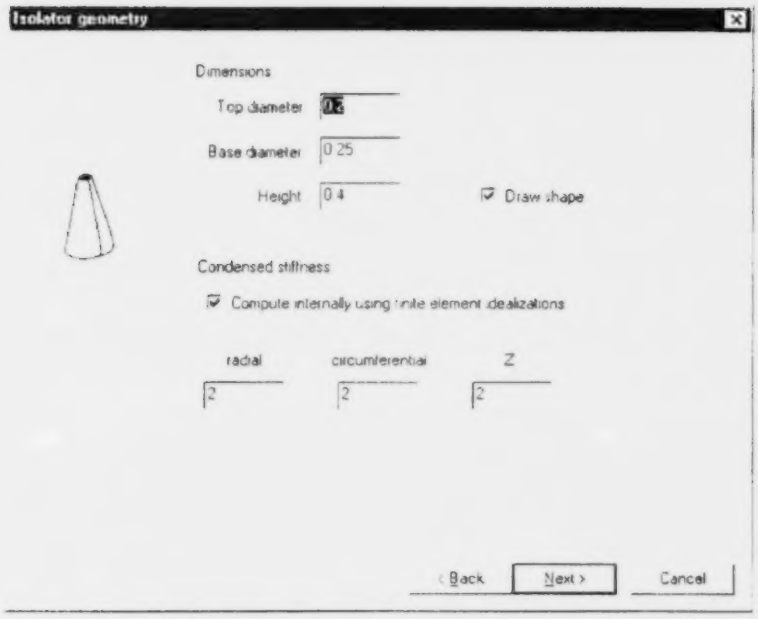

Figure 7. Model Wizard: Solid Cone Isolator Geometry

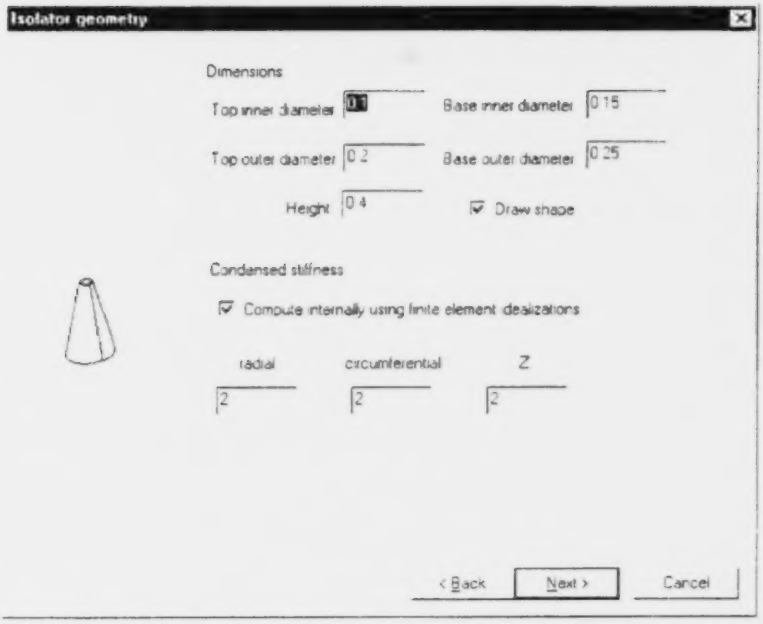

Figure 8. Model Wizard: Hollow Cone Isolator Geometry

P517368.PDF [Page: 33 of 68]

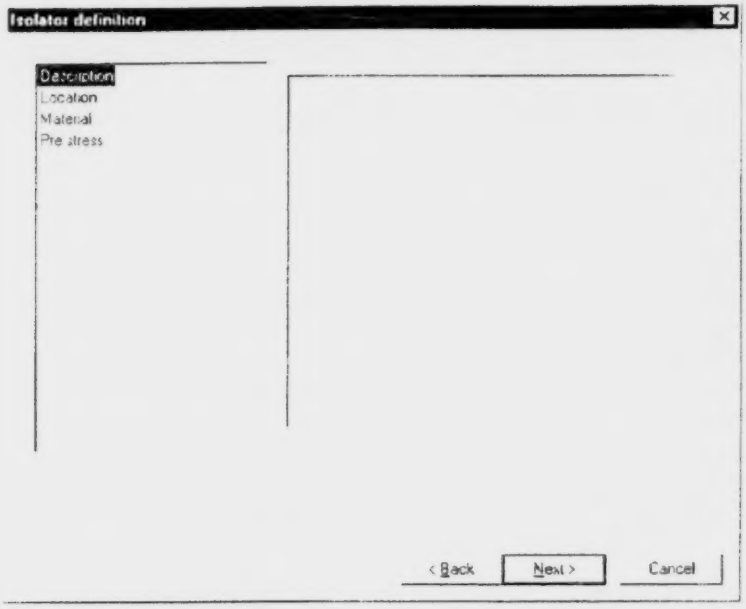

Figure 9. Model Wizard: Isolator Definition – Description

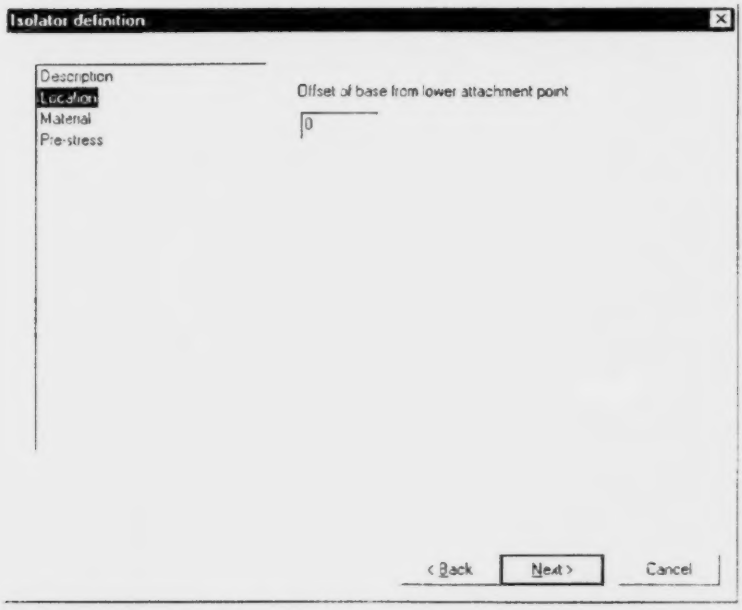

Figure 10. Model Wizard: Isolator Definition – Location

P517368 PDF [Page: 34 of 68]

Figure 11. Model Wizard: Isolator Definition – Material

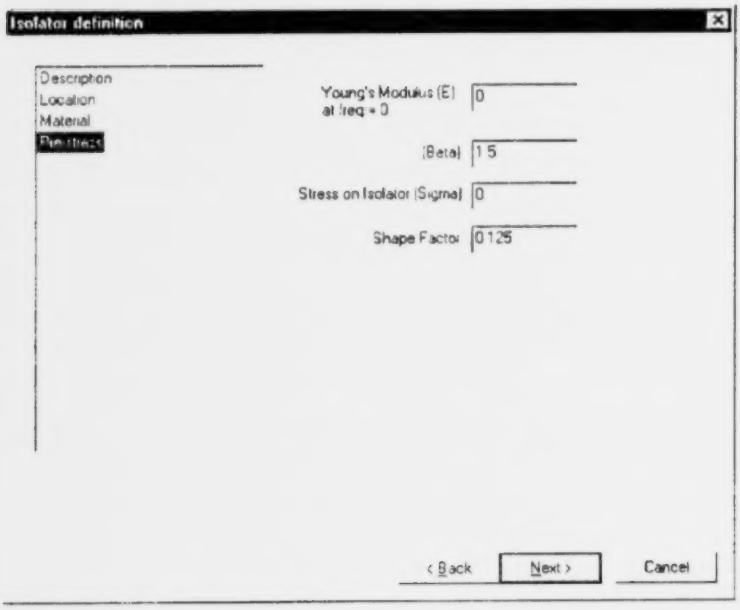

Figure 12. Model Wizard: Isolator Definition – Pre-stress

P517368.PDF [Page: 35 of 68]

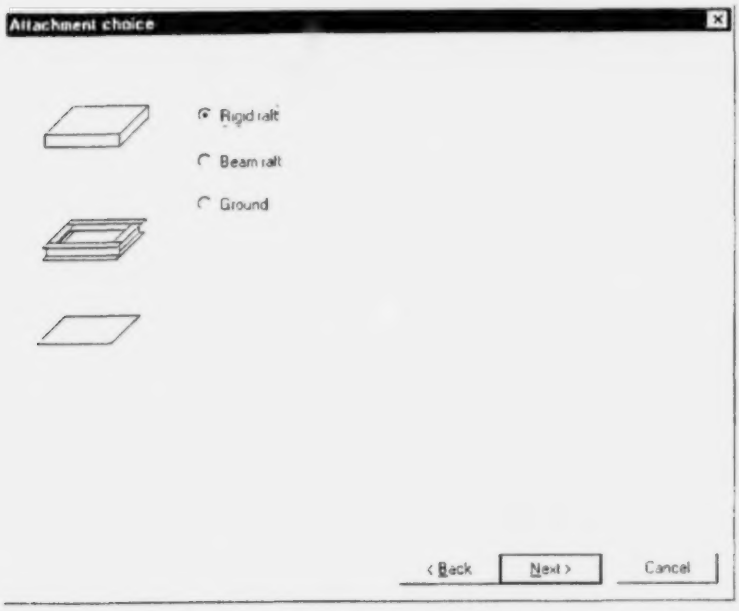

Figure 13. Model Wizard: Attachment Choice

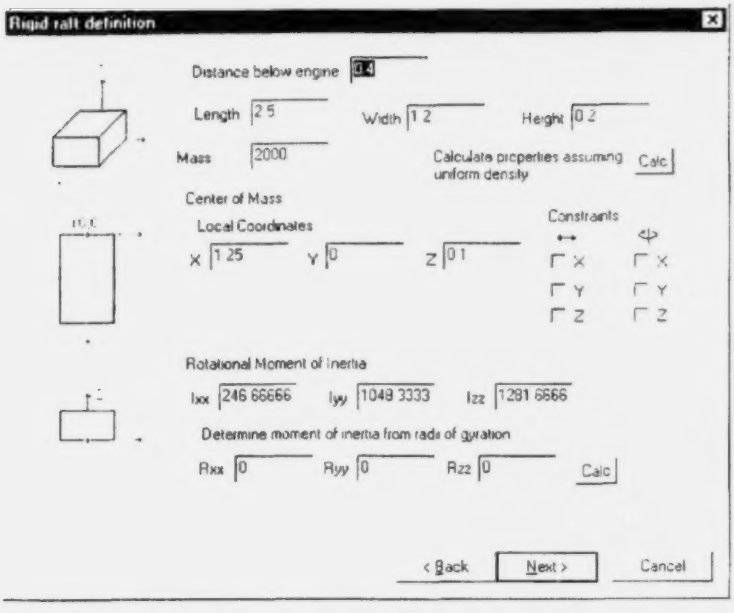

Figure 14. Model Wizard: Rigid Raft Definition

P517368.PDF [Page: 36 of 68]

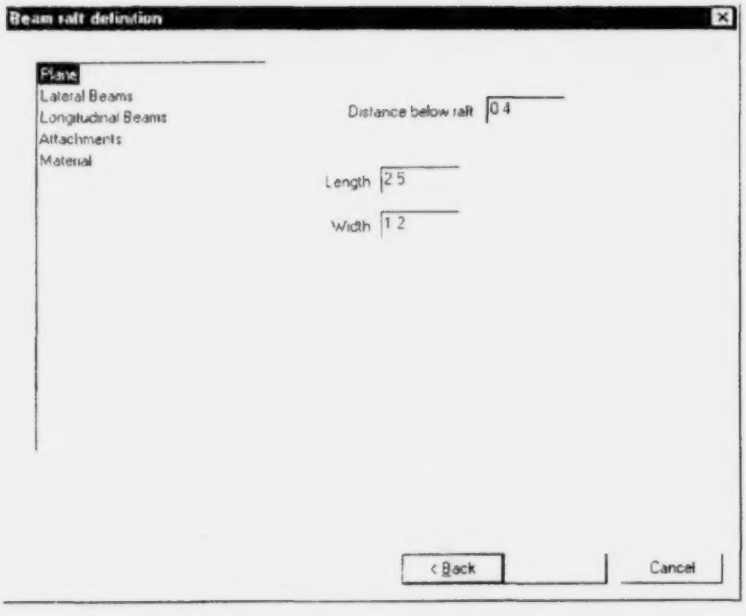

Figure 15. Model Wizard: Beam Raft – Plane Definition

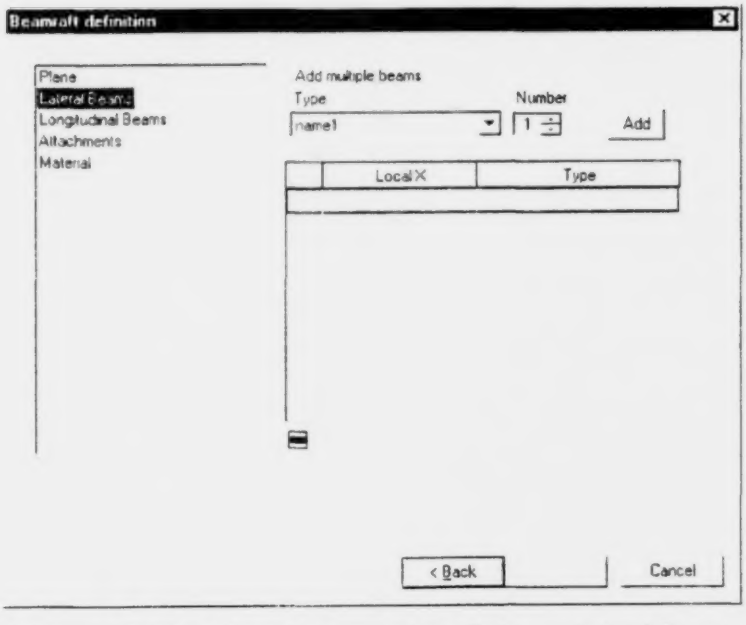

Figure 16. Model Wizard: Beam Raft Definition – Lateral Beams

P517368.PDF [Page: 37 of 68]

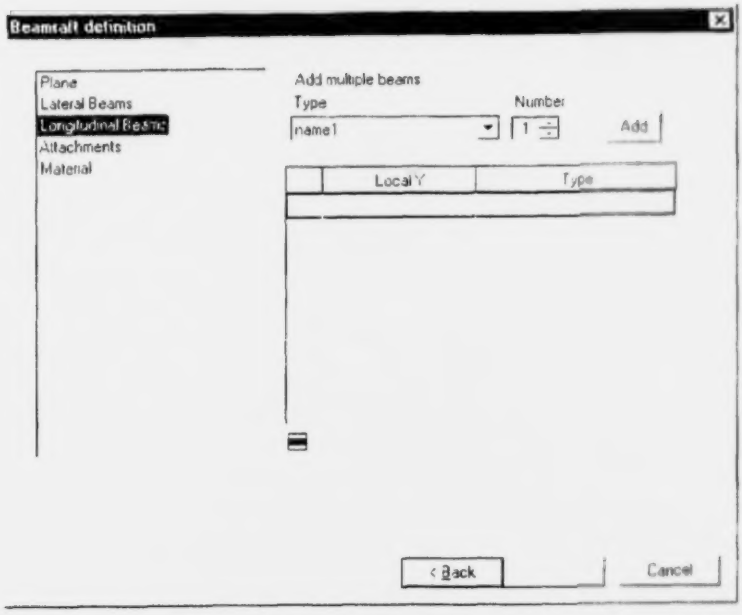

Figure 17. Model Wizard: Beam Raft Definition – Longitudinal Beams

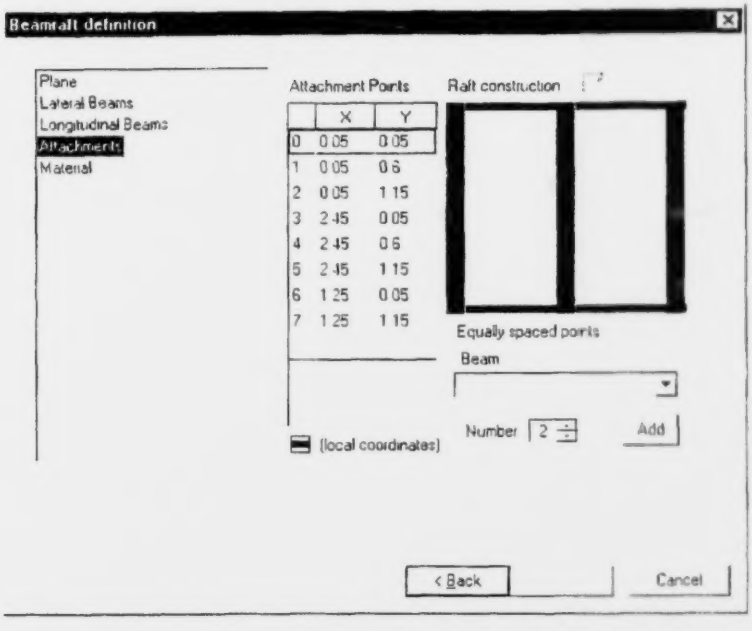

Figure 18. Model Wizard: Beam Raft Definition – Attachments

P517368.PDF [Page: 38 of 68]

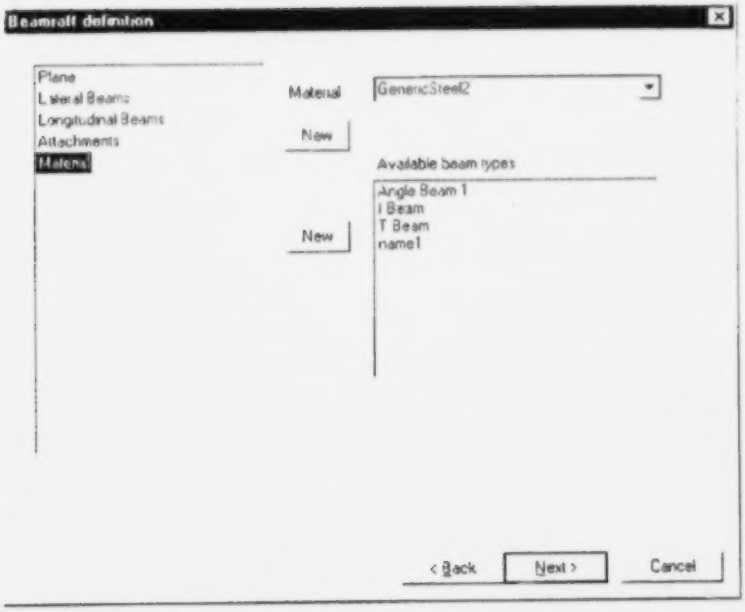

Figure 19. Model Wizard: Beam Raft Definition – Material

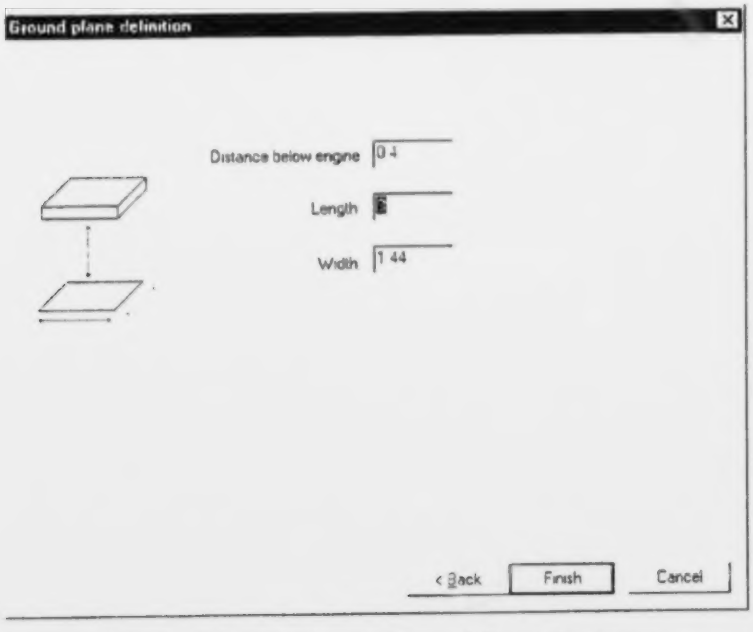

Figure 20. Model Wizard: Ground Plane Definition

P517368.PDF [Page: 39 of 68]

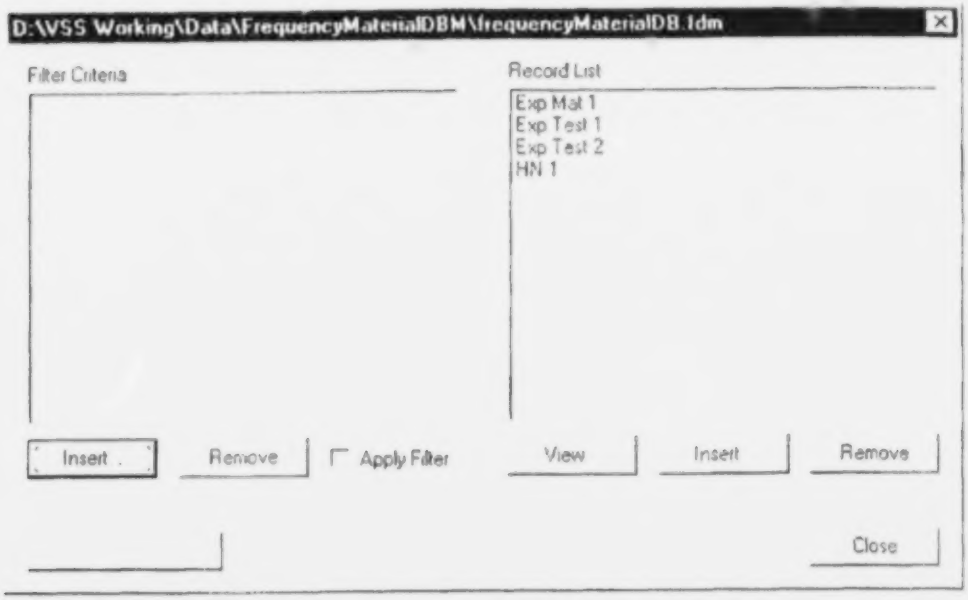

Figure 21. Frequency Dependent Material Database Manager

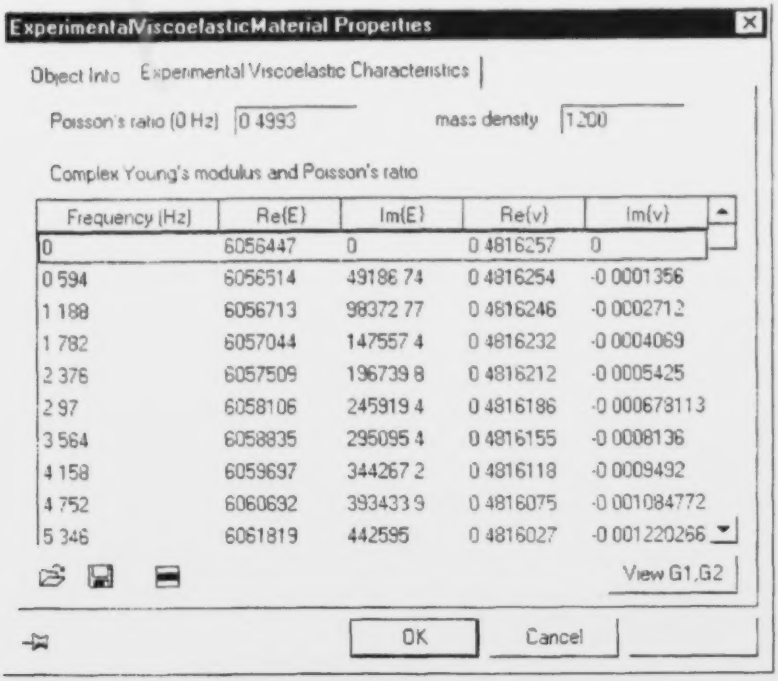

Figure 22. Experimental Viscoelastic Material Dialog

P517368.PDF [Page: 40 of 68]

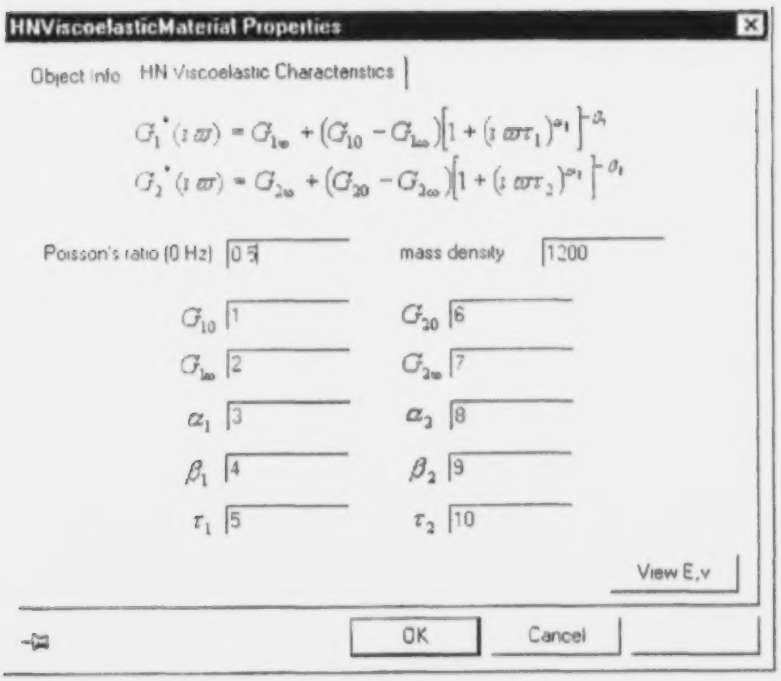

Figure 23. Havrilak Negami Viscoelastic Material Dialog

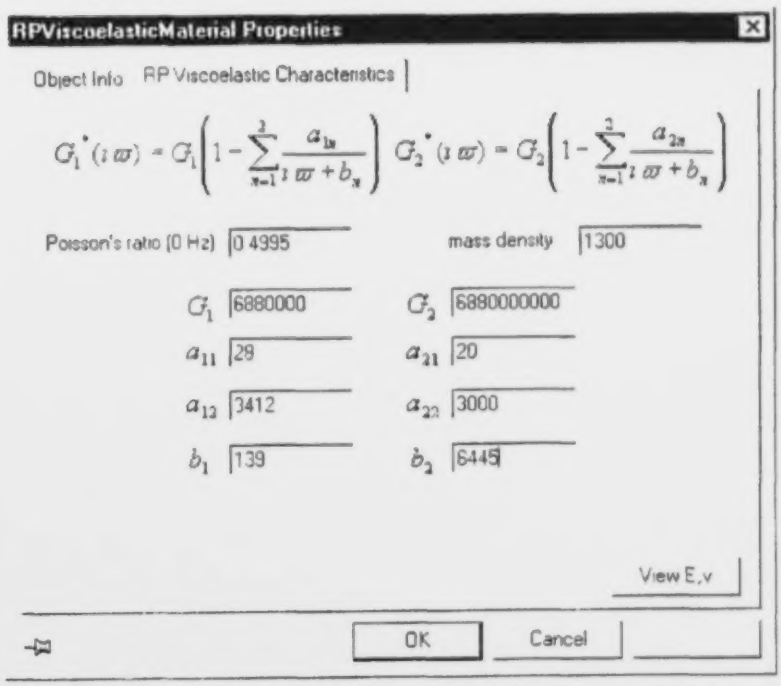

Figure 24. Rational Polynomial Viscoelastic Material Dialog

P517368.PDF [Page: 41 of 68]

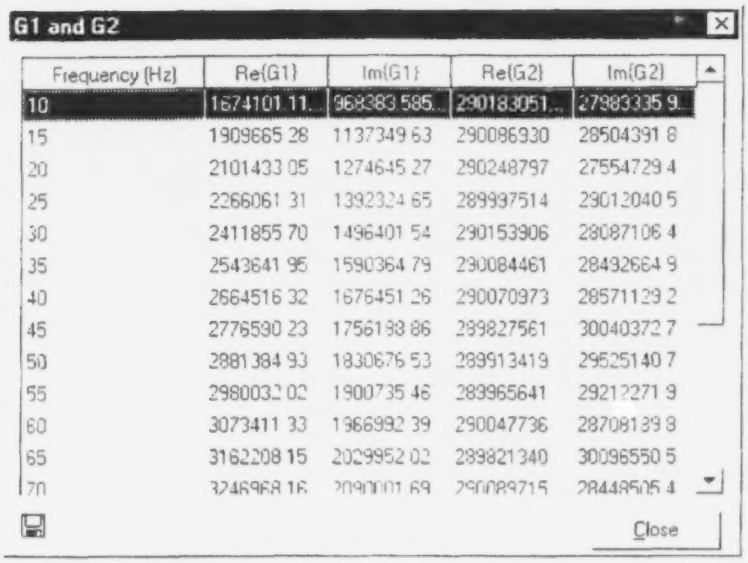

Figure 25. G1 and G2 Dialog

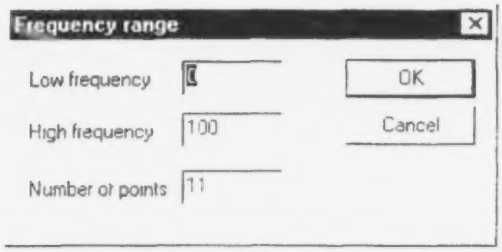

Figure 26. Frequency Range Dialog

P517368.PDF [Page: 42 of 68]

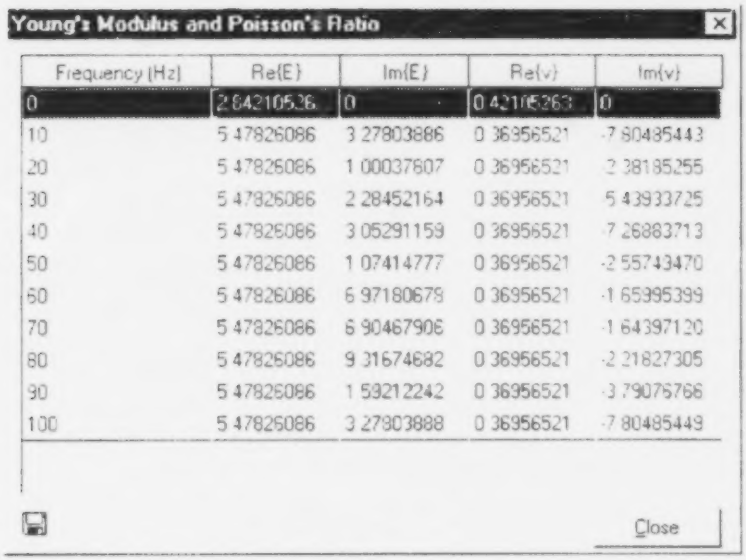

Figure 27. Young's Modulus and Poisson's Ratio Dialog

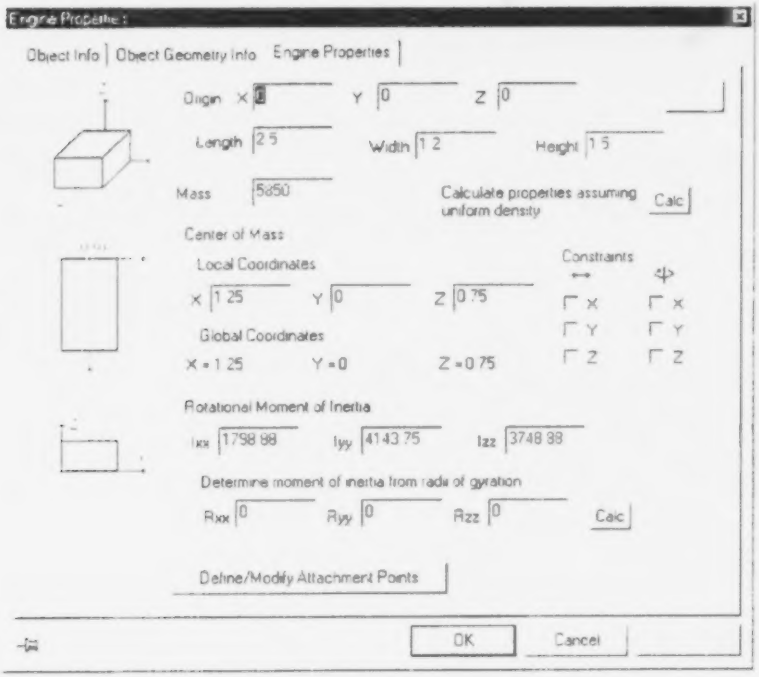

Figure 28. Engine Properties Dialog

P517368.PDF [Page: 43 of 68]

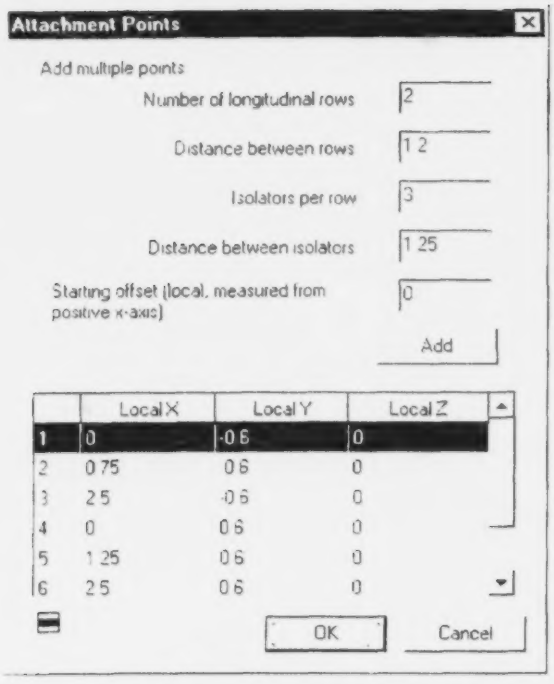

Figure 29. Attachment Points Dialog

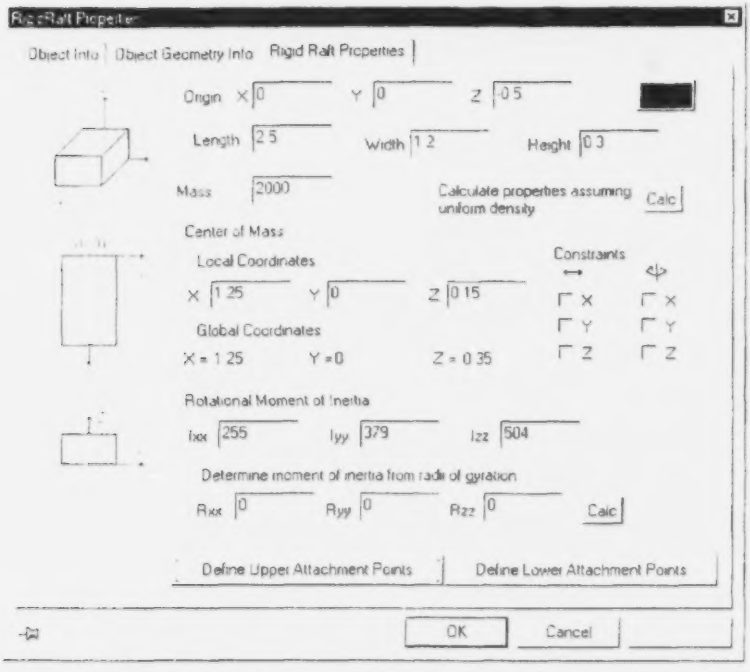

Figure 30. Rigid Raft Properties Dialog

P517368.PDF [Page: 44 of 68]

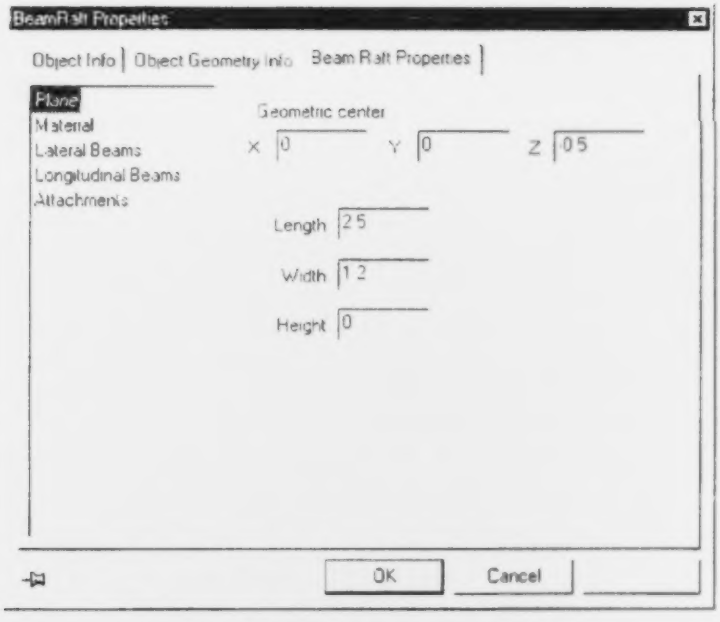

Figure 31. Beam Raft Properties Dialog

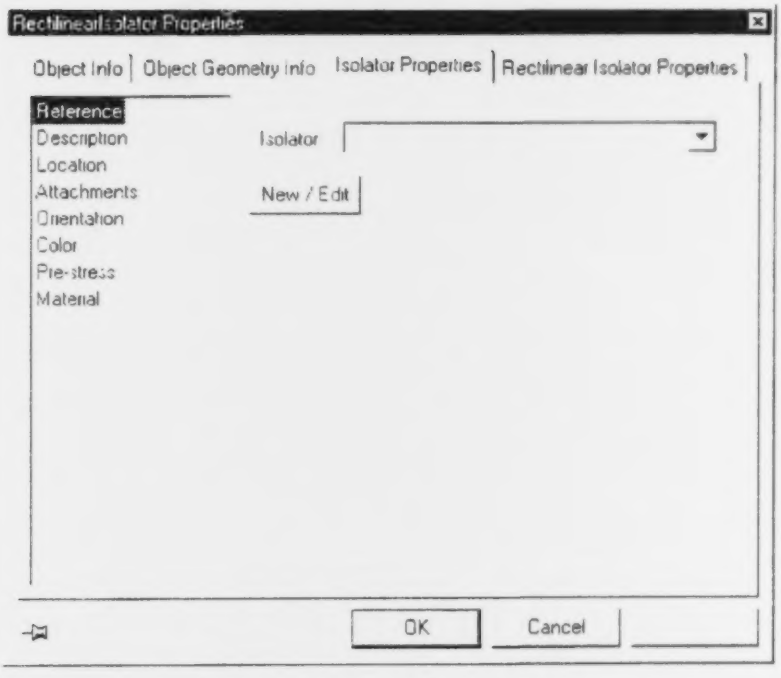

Figure 32. Isolator Reference Dialog

P517368.PDF [Page: 45 of 68]

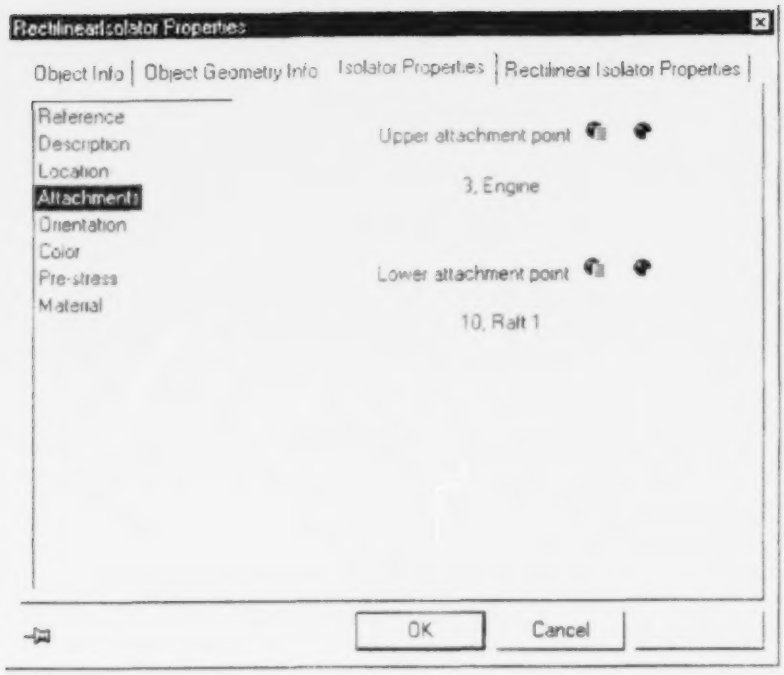

Figure 33. Isolator Attachments Dialog

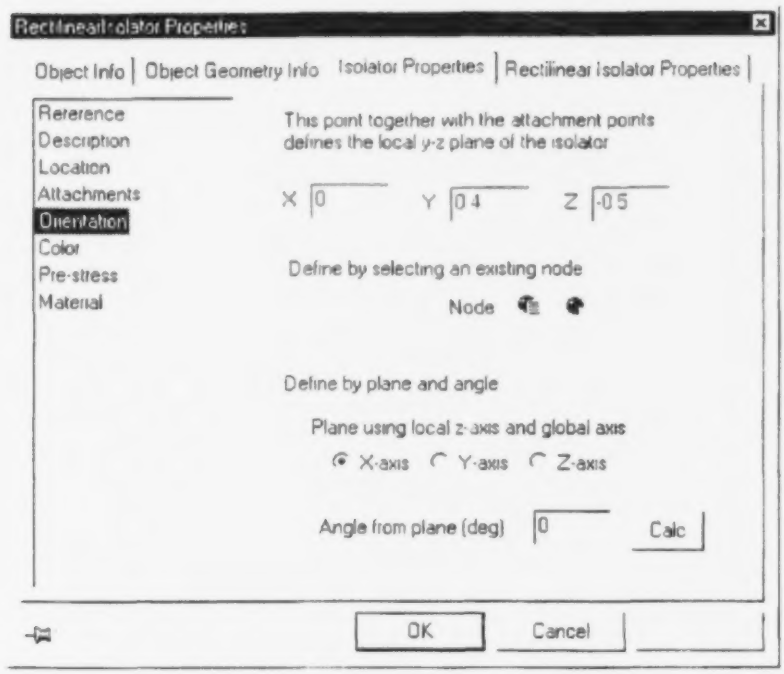

Figure 34. Isolator Orientation Dialog

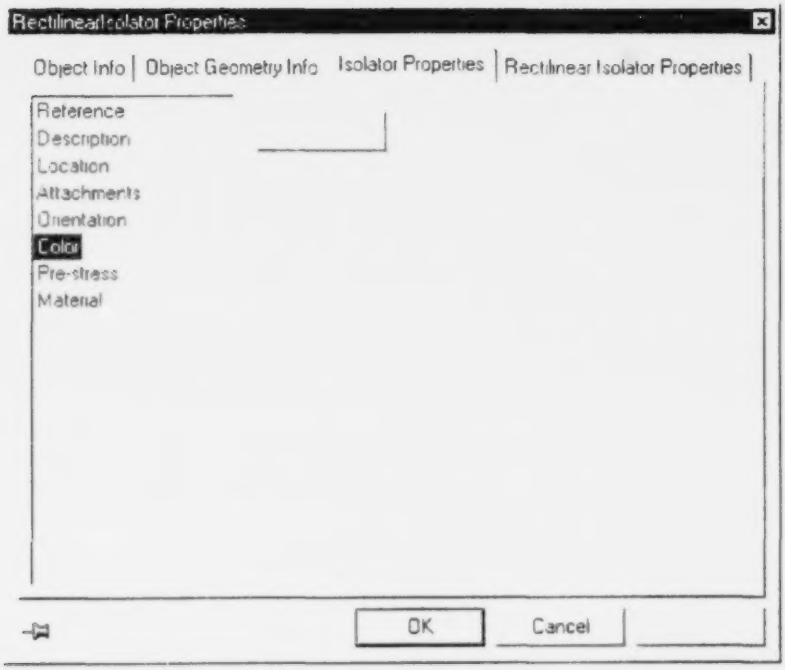

Figure 35. Isolator Color Dialog

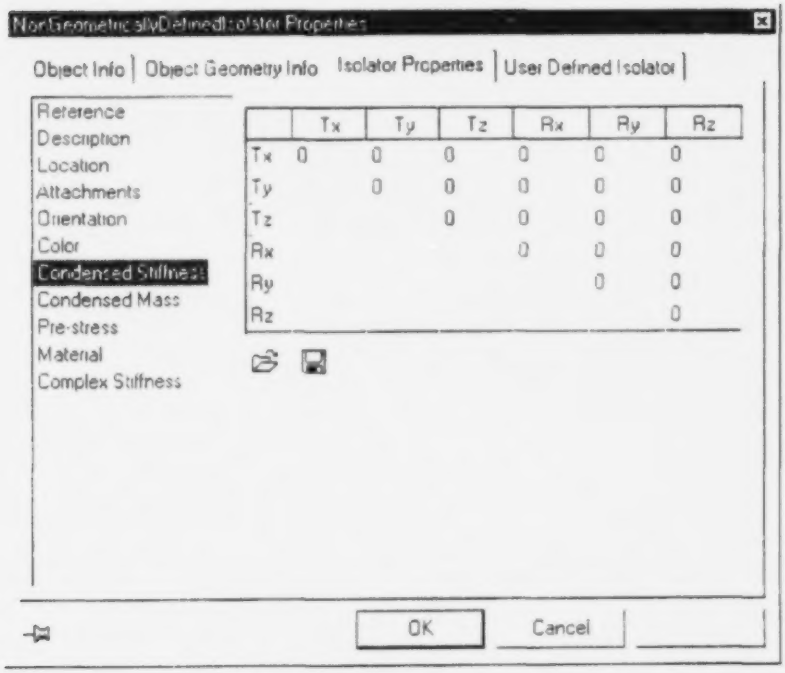

Figure 36. Isolator Condensed Stiffness Dialog

P517368.PDF [Page: 47 of 68]

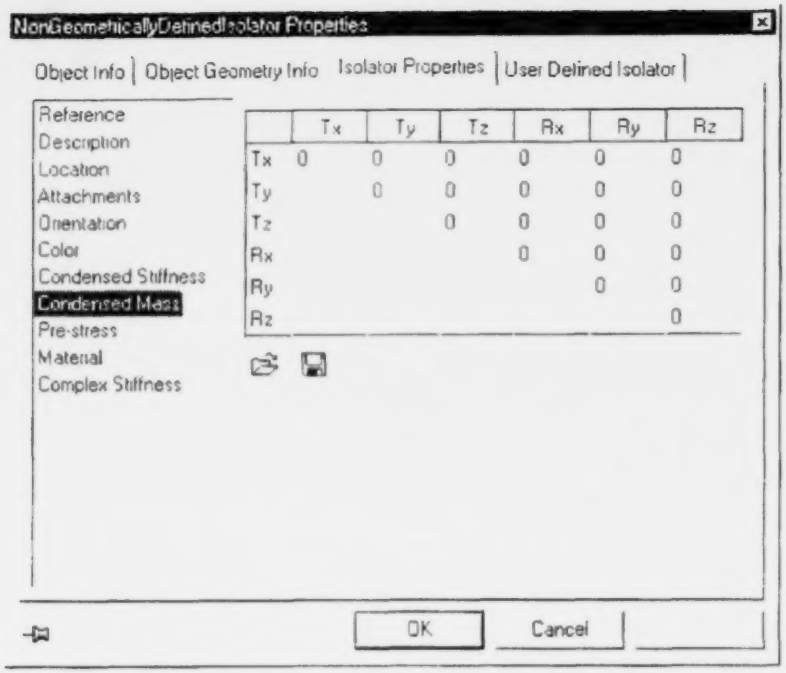

Figure 37. Isolator Condensed Mass Dialog

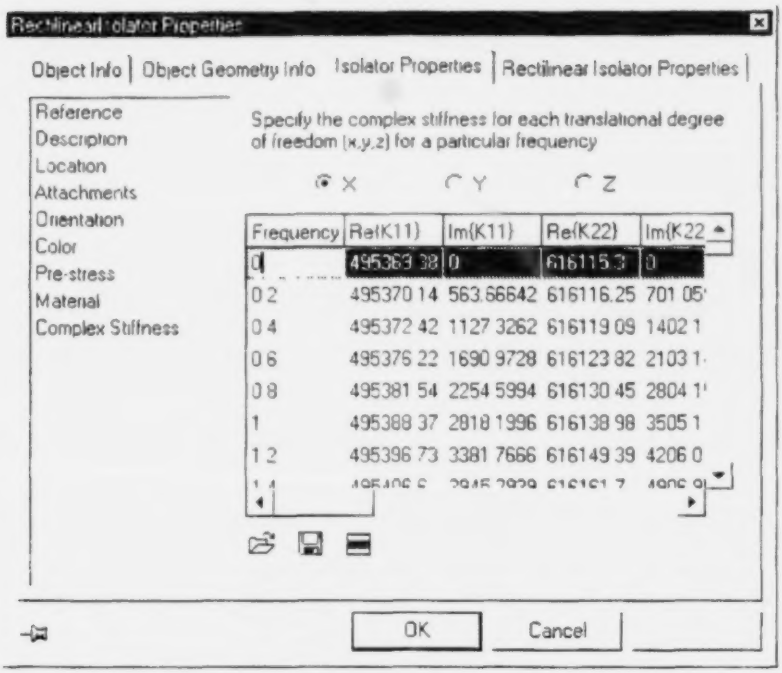

Figure 38. Isolator Complex Stiffness Dialog

P517368.PDF [Page: 48 of 68]

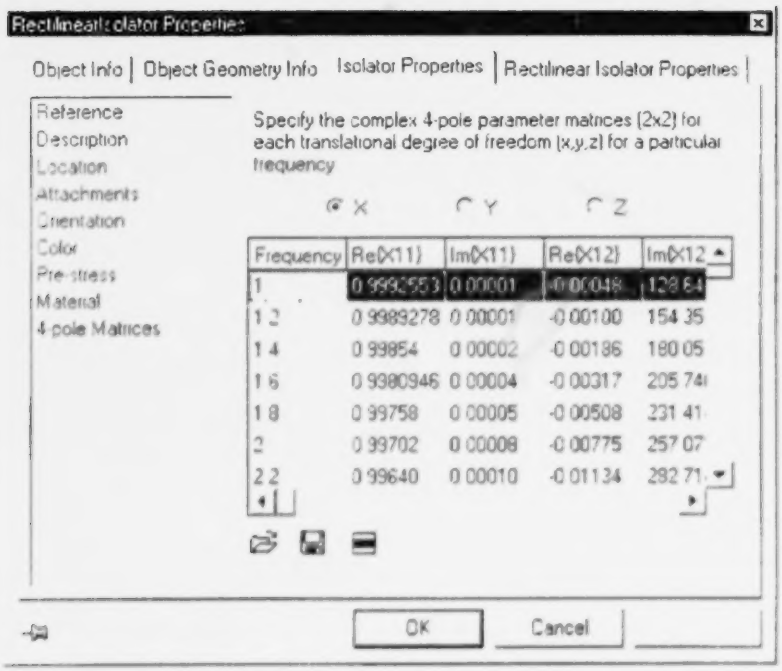

Figure 39. Isolator 4-pole Matrices Dialog

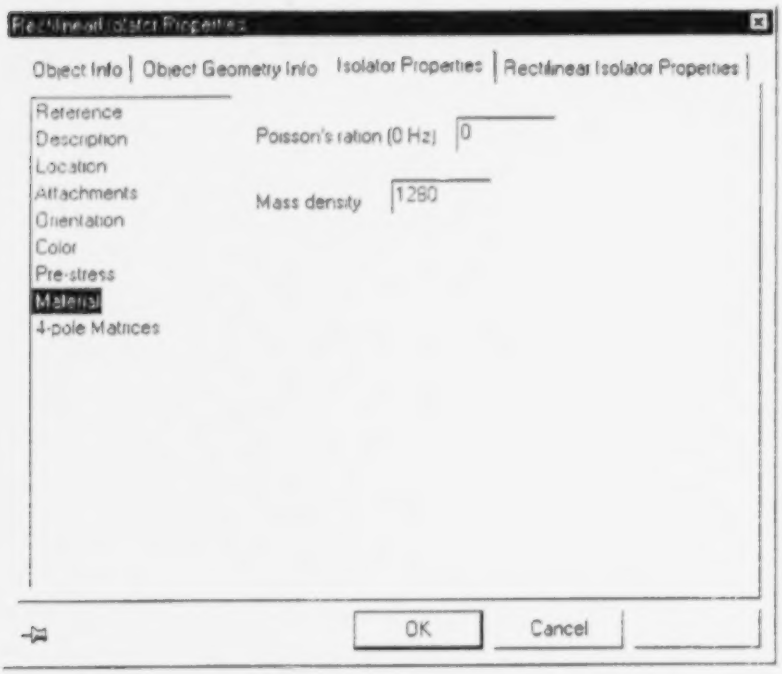

Figure 40. Isolator Material Dialog (DYNSTFF,4-POLE)

P517368.PDF [Page: 49 of 68]

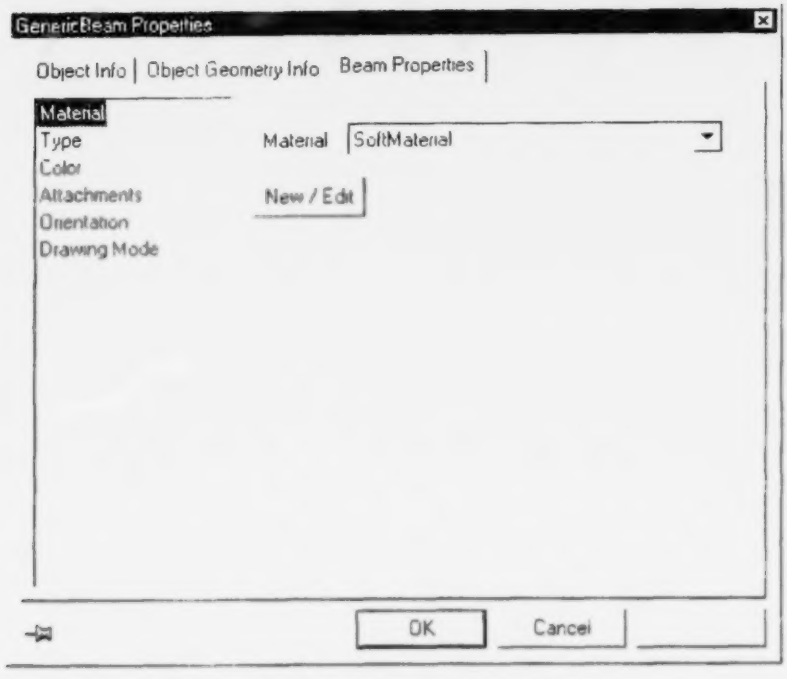

Figure 41. Generic Beam Material Dialog

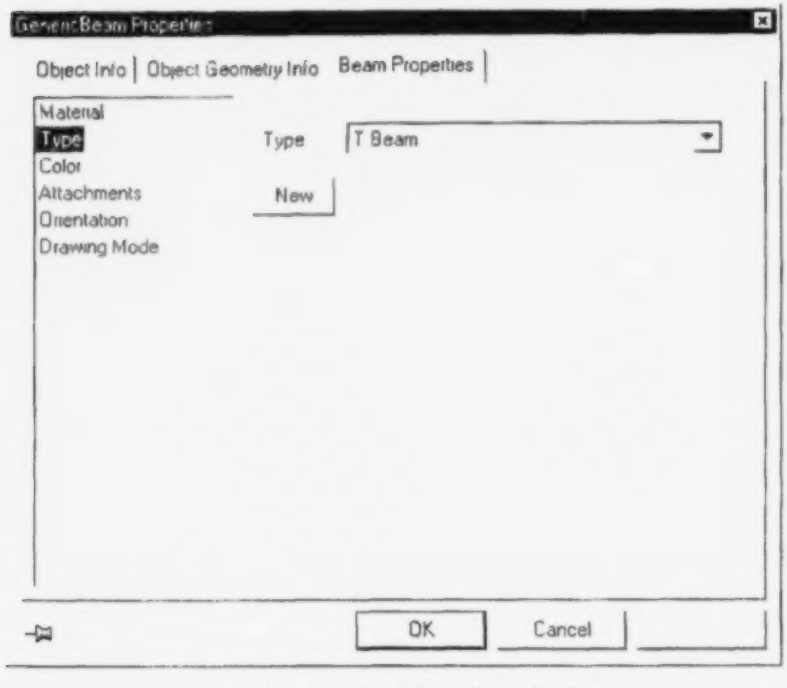

Figure 42. Generic Beam Type Dialog

P517368.PDF [Page: 50 of 68]

Figure 43. Generic Beam Attachments Dialog

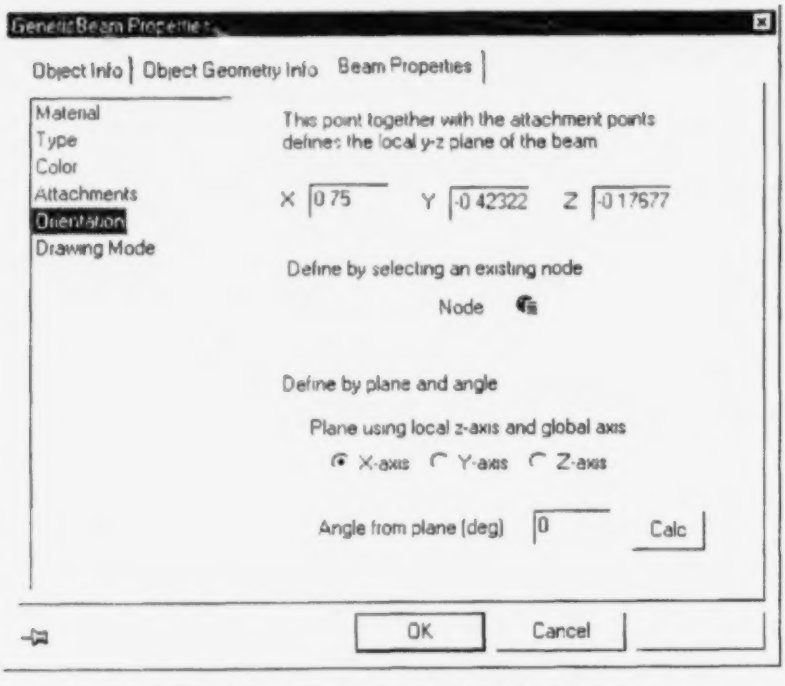

Figure 44. Generic Beam Orientation Dialog

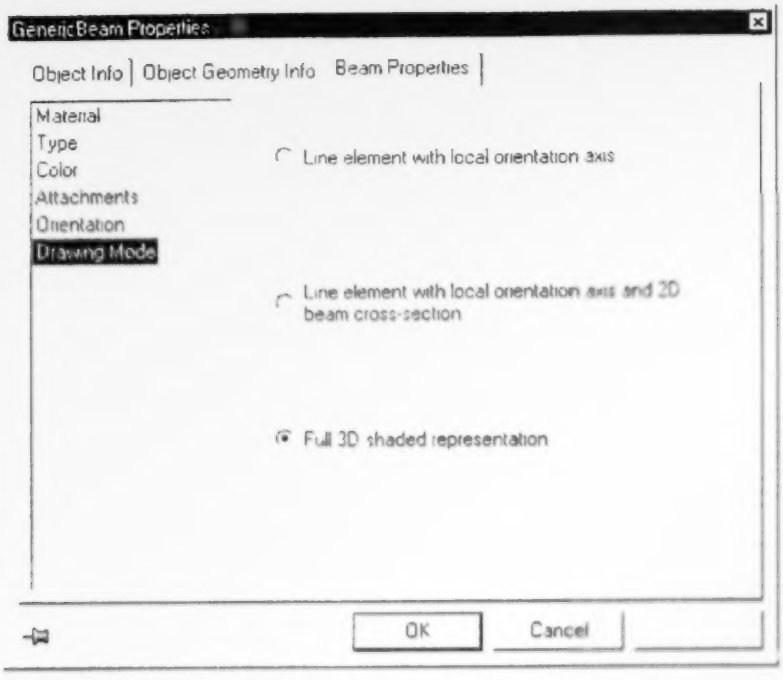

Figure 45. Generic Beam Drawing Mode Dialog

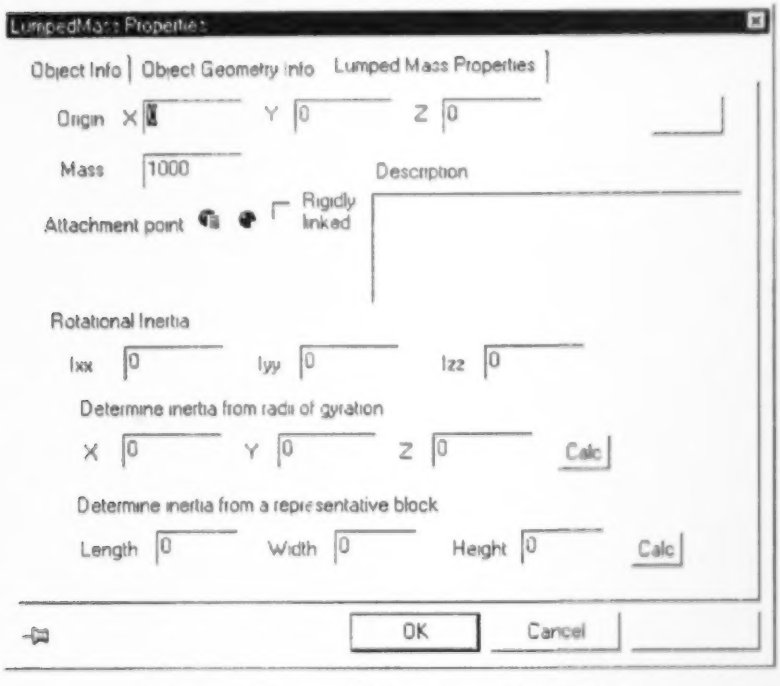

Figure 46. Lumped Mass Properties Dialog

Figure 47. Attachment Point Properties Dialog

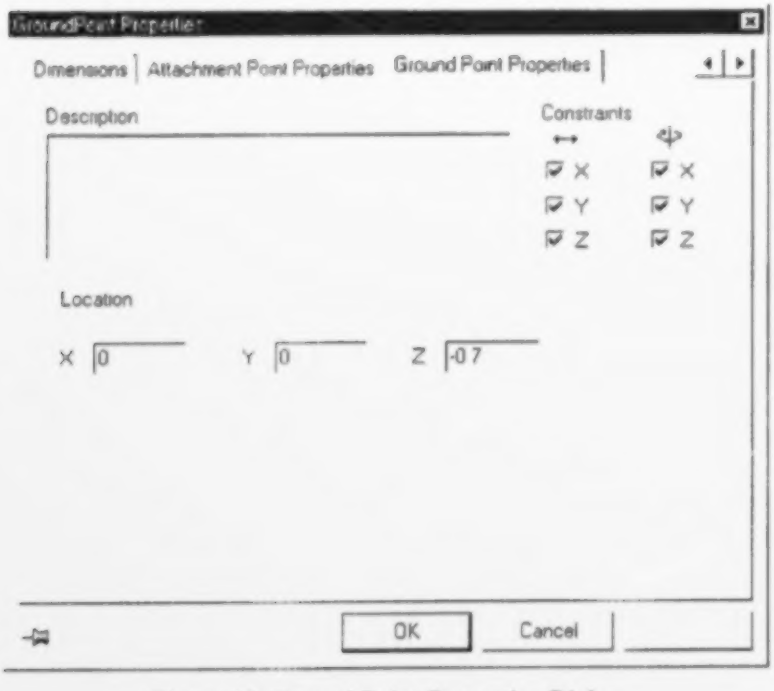

Figure 48. Ground Point Properties Dialog

P517368.PDF [Page 53 of 68]

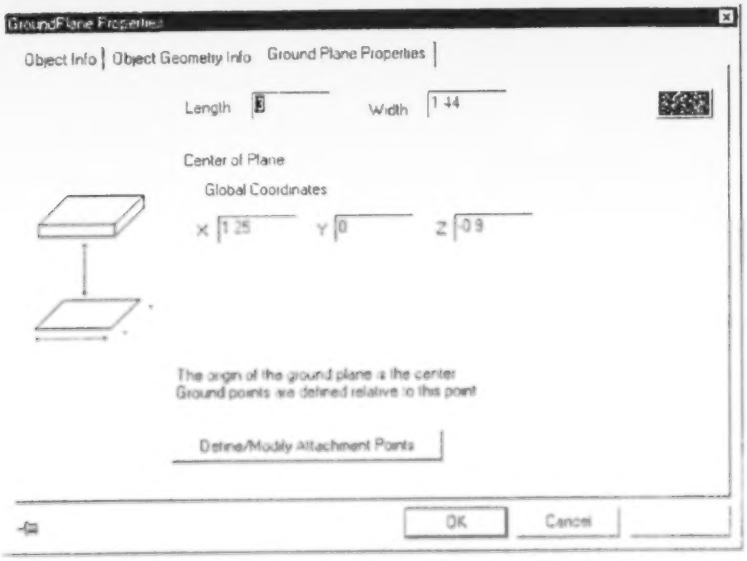

Figure 49. Ground Plane Properties Dialog

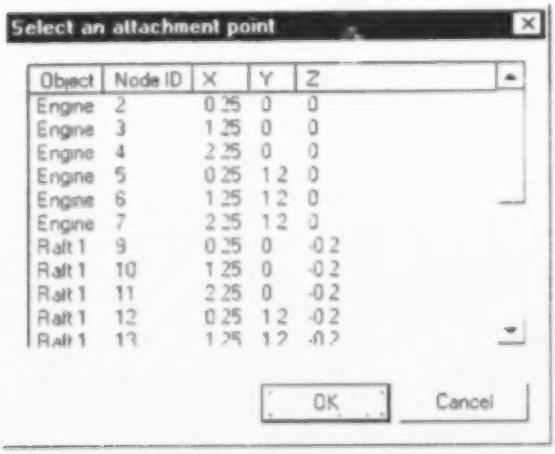
Select an attachment point
Object
Node ID
X
Y
Z
Engine 2 0 25 0 0
Engine 3 1 25 0 0
Engine 4 2 25 0 0
Engine 5 0 25 1 2 0
Engine 6 1 25 1 2 0
Engine 7 2 25 1 2 0
Raft 1 9 0 25 0 -0 2
Raft 1 10 1 25 0 -0 2
Raft 1 11 2 25 0 -0 2
Raft 1 12 0 25 1 2 -0 2
OK
Cancel

Figure 50. Attachment Point Picking Dialog

P517368.PDF [Page: 54 of 68]

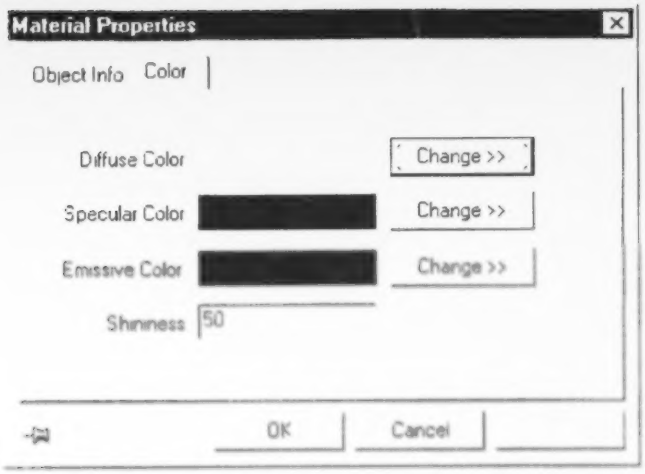

Figure 51. Color Modification Dialog

P517368.PDF [Page: 55 of 68]

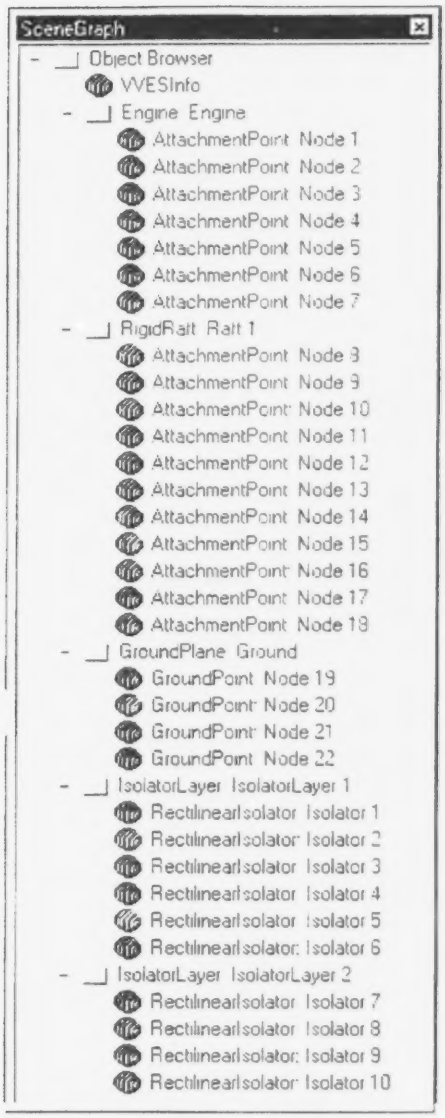

Figure 52. Scene Graph Tree View

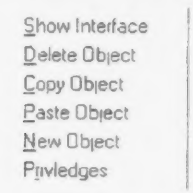

Figure 53. Scene Graph Tree Popup Menu

P517368.PDF [Page 36 of 68]

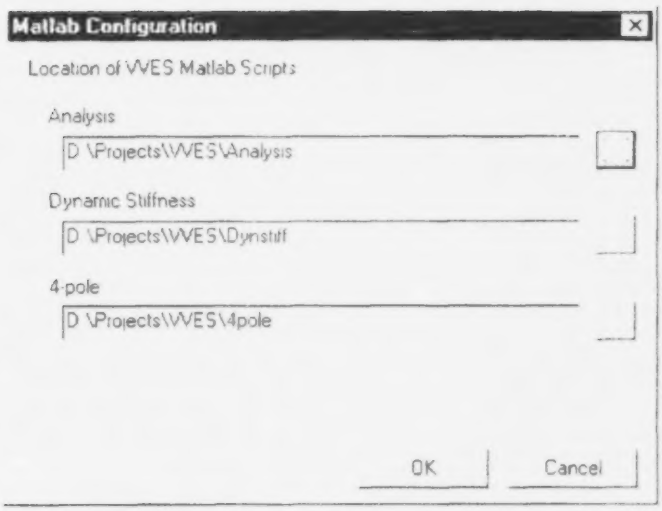

Figure 54. Matlab Configuration Dialog

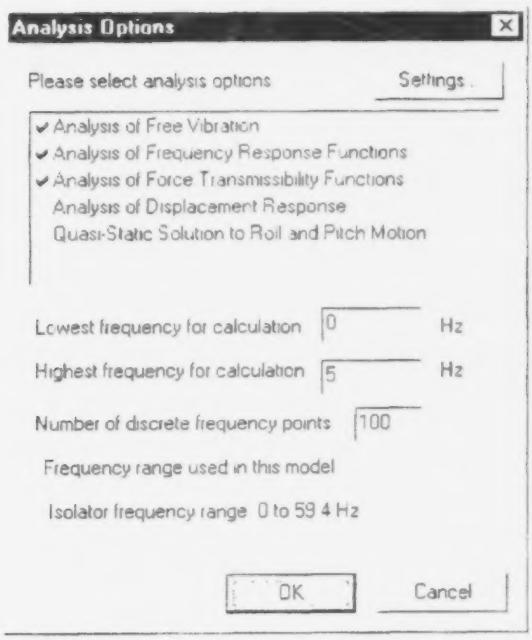

Figure 55. Analysis Options Dialog

P517368 PDF [Page: 57 of 68]

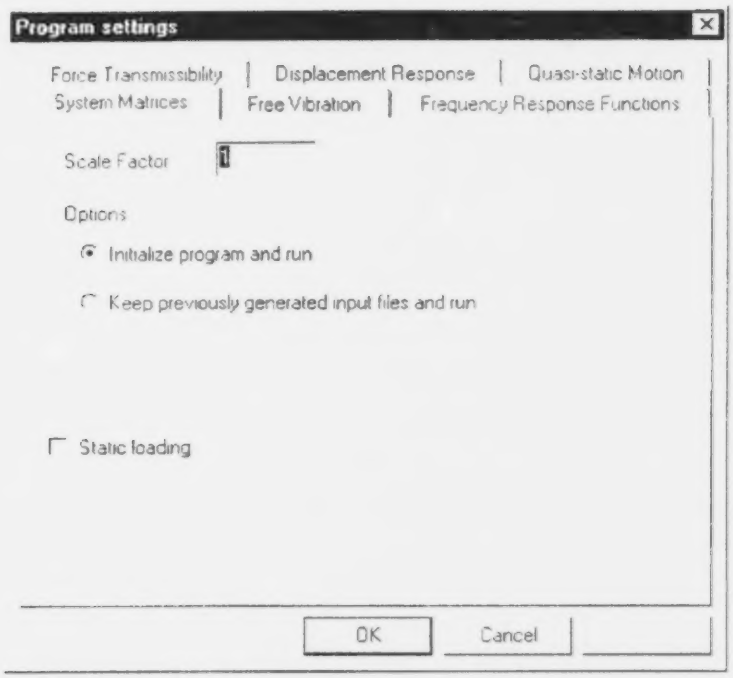

Figure 56. System Matrices Settings

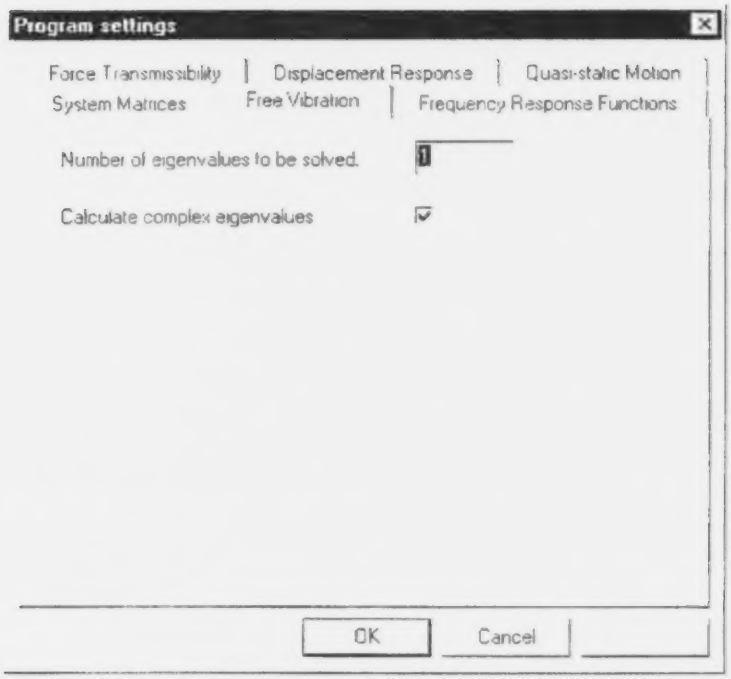

Figure 57. Free Vibration Settings

P517368.PDF [Page: 58 of 68]

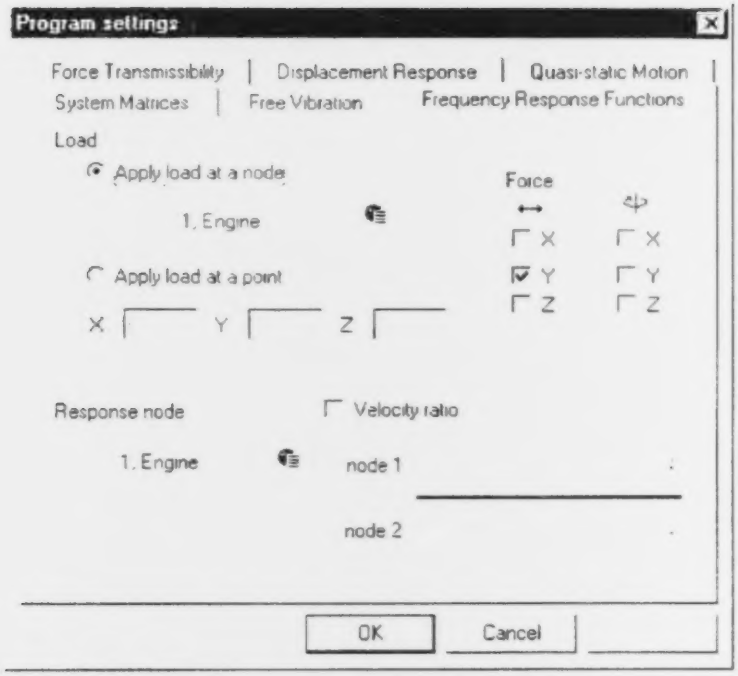

Figure 58. Frequency Response Functions Settings

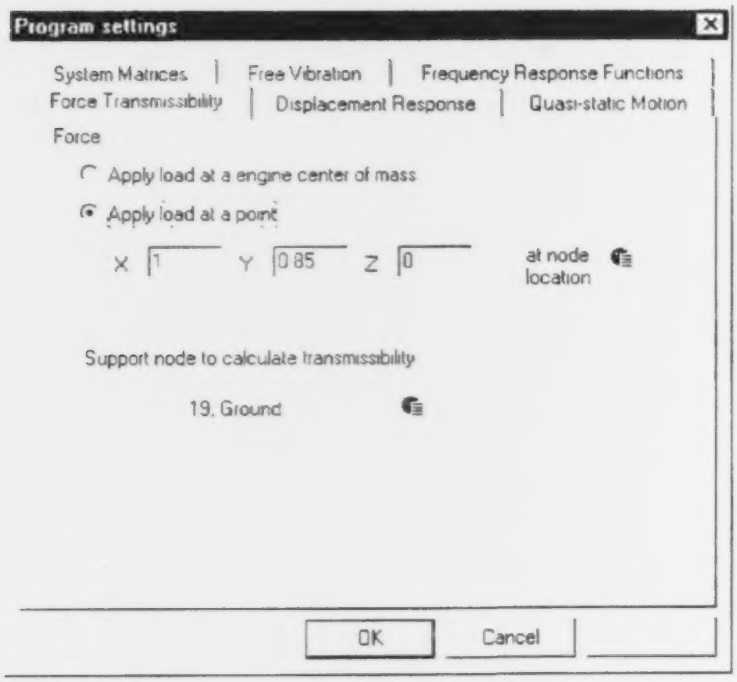

Figure 59. Force Transmissibility Settings

P517368.PDF [Page: 59 of 68]

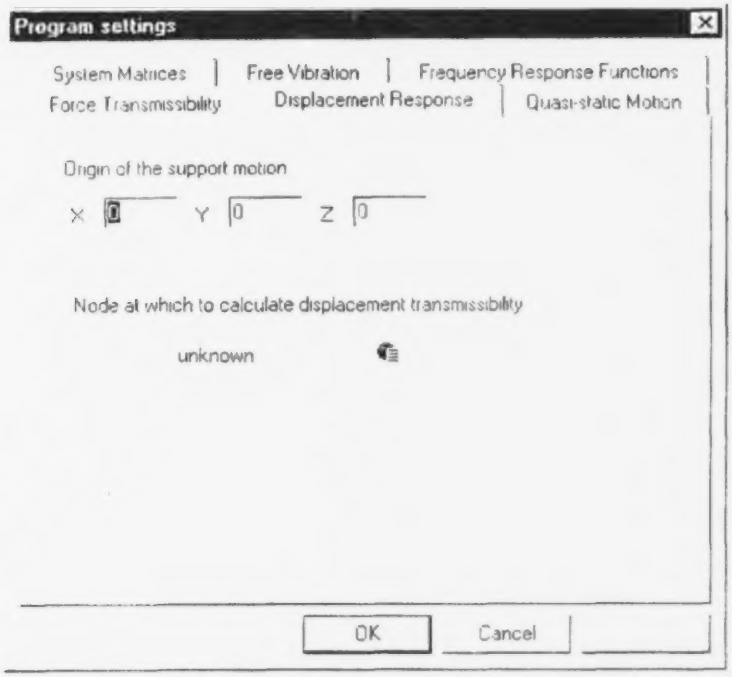

Figure 60. Displacement Response Settings

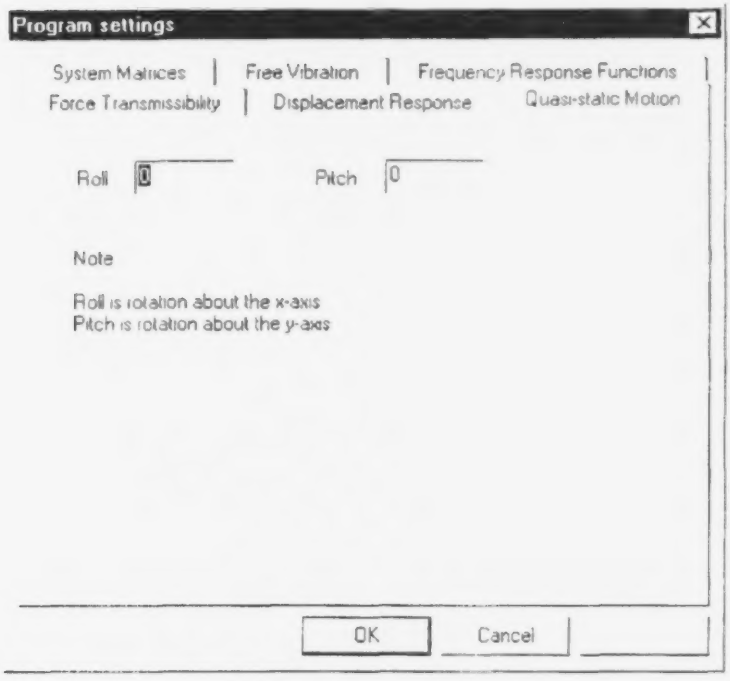

Figure 61. Quasi-static Motion Settings

P517368.PDF [Page: 60 of 68]

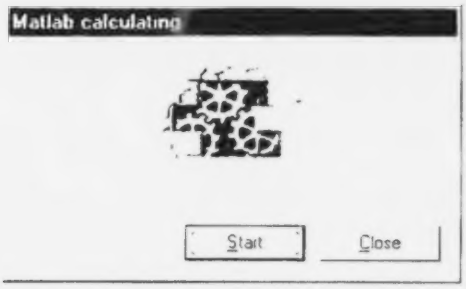

Figure 62. Calculation Dialog

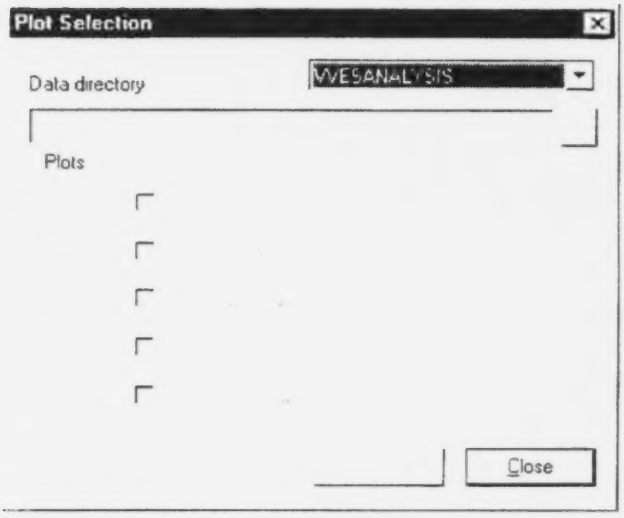

Figure 63. Adding Plots

Object Info
ChartBrowser
Acceleration FRF Excited Node 1 (Fy), Response Node 1
Velocity FRF Excited Node 1 (Fy), Response Node 1
Displacement FRF Excited Node 1 (Fy), Response Node 1
Show Chart
OK
Cancel

P517368.PDF [Page: 61 of 68]

Figure 64. Selecting a Chart for Display

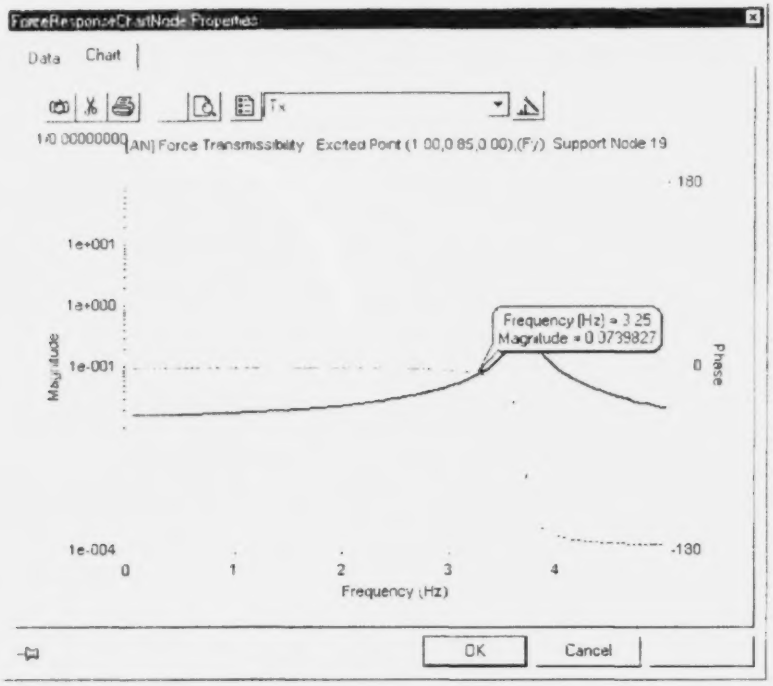

Figure 65. Sample Chart

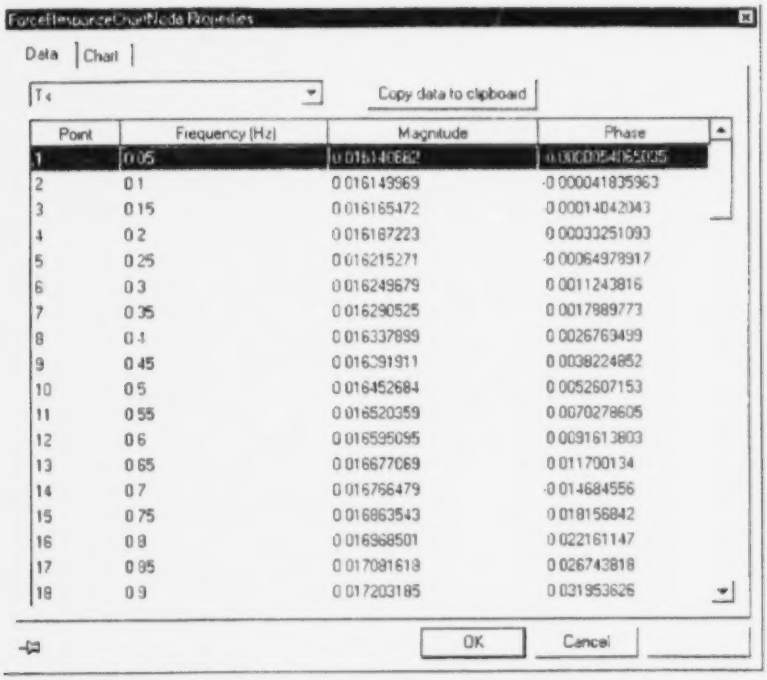

Figure 66. Sample Chart Data

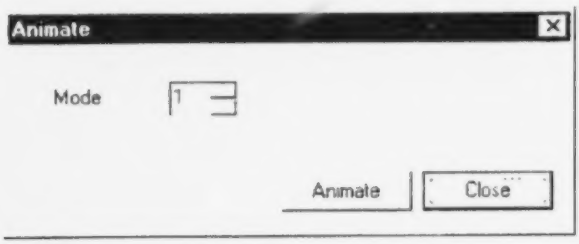

Figure 67. Animation Dialog

P517368.PDF [Page: 63 of 68]

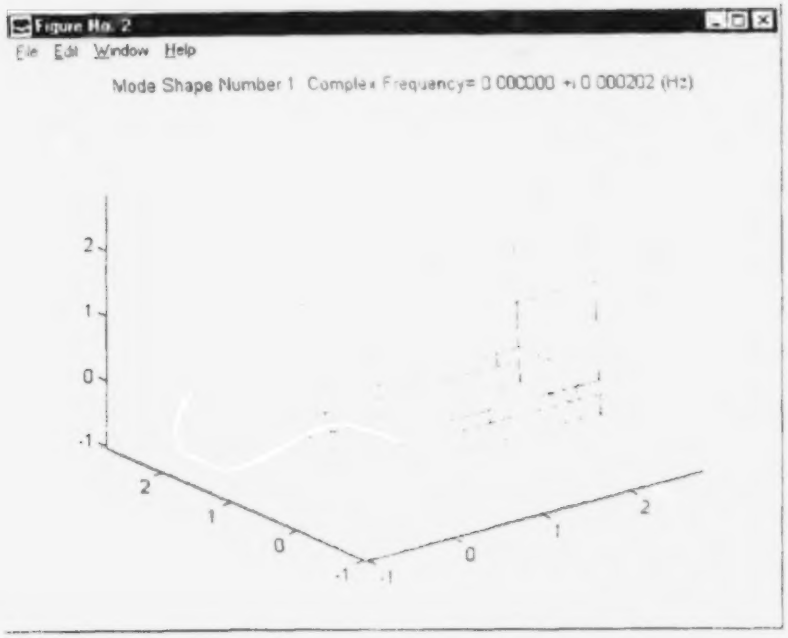

Figure 68. Animation in Progress

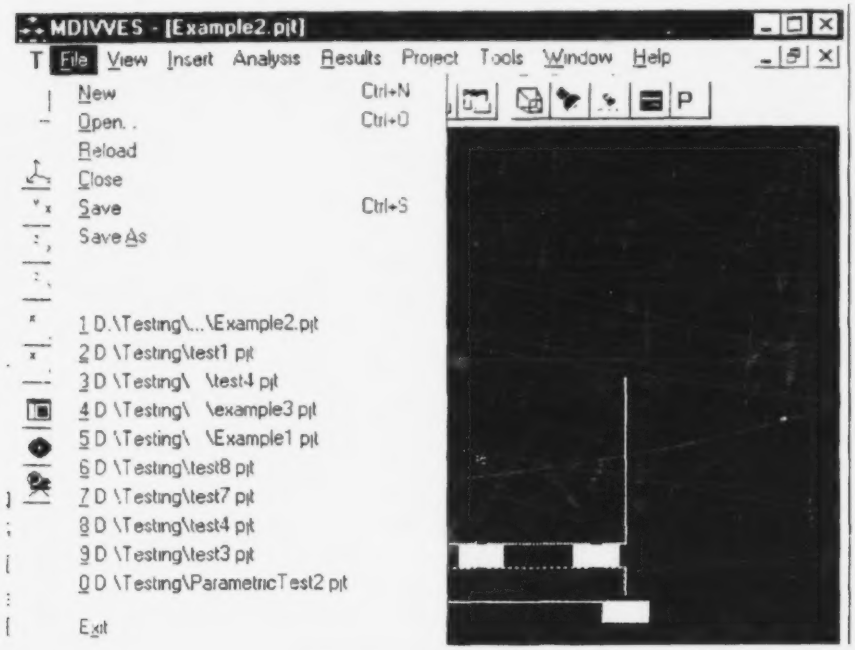

Figure 69. The File Menu

P517368 PDF [Page: 64 of 68]

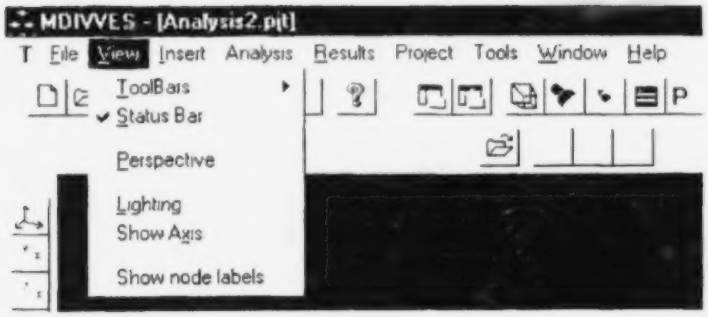

Figure 70. The View Menu

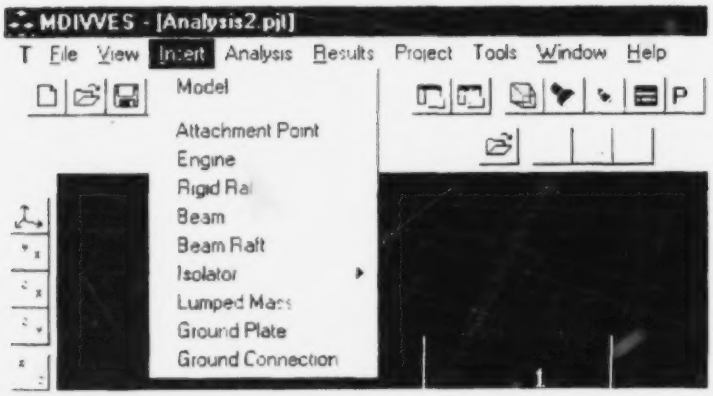

Figure 71. The Insert Menu

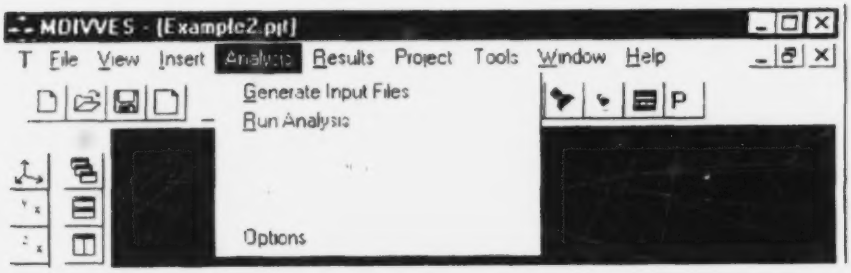

Figure 72. The Analysis Menu

P517368.PDF [Page: 65 of 68]

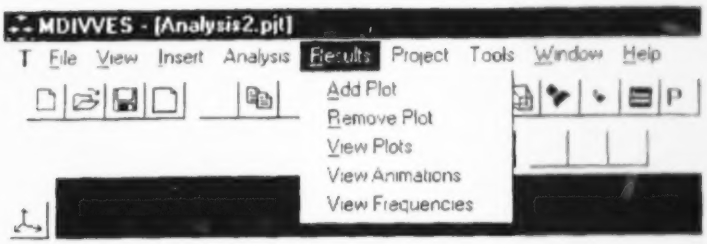

Figure 73. The Results Menu

Figure 74. The Project Menu

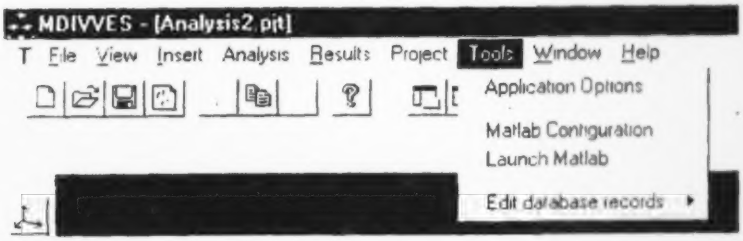

Figure 75. The Tools Menu

P517368.PDF [Page: 66 of 68]

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1 ORIGINATOR (the name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.)

MARTEC Limited
1888 Brunswick Street, Suite 400,
Halifax, Nova Scotia, Canada, B3J 3J8

2 SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable)

UNCLASSIFIED

3 TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title)

VVES Model Generator and Graphical User Interface Application Version 1.0 User's Manual

4 AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.)

Polak, M.F., Jiang, L.

5 DATE OF PUBLICATION (month and year of publication of document)	6a NO. OF PAGES (total containing information. Include Annexes, Appendices, etc)	6b NO. OF REFS (total cited in document)
January 2002	50	1

7 DESCRIPTIVE NOTES (the category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered)

CONTRACTOR REPORT

8 SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address)

Defence Research Establishment Atlantic
PO Box 1012
Dartmouth, NS, Canada B2Y 3Z7

9a PROJECT OR GRANT NO. (if appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant)	9b CONTRACT NO. (if appropriate, the applicable number under which the document was written)
Project 1gc12	W7707- 008286/001/HAL

10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity. This number must be unique to this document.)	10b OTHER DOCUMENT NOs. (Any other numbers which may be assigned this document either by the originator or by the sponsor.)
Martec Report No. TR-01-21	DREA CR 2001-142

11 DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(X) Unlimited distribution
( ) Defence departments and defence contractors; further distribution only as approved
( ) Defence departments and Canadian defence contractors; further distribution only as approved
( ) Government departments and agencies; further distribution only as approved
( ) Defence departments; further distribution only as approved
( ) Other (please specify):

12. DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected).

Unlimited

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual)

The VVES (Vibration of Visco-elastic and Elastic Structures) suite of programs is a group of Matlab codes for analysis and design of engine isolation systems composed of engine, raft and ground which are connected by visco-elastic isolators. Among these programs, VVESANALYSIS allows the dynamic characteristics of isolators to be described by frequency-dependent complex material properties. The VVESDYNSTIFF and VVES4POLE require input of the directly measured dynamic stiffnesses and the 4 pole parameter matrices, respectively.

In order to simplify the construction of input files and provide an alternate method to view the VVES analysis results, a model generator and graphical user interface for the VVES programs has been recently developed. This model generator and GUI is developed for the Windows environment using the HOOD toolkit. This user's manual provides instructions on using the VVES model generator and GUI. The reader of this report is assumed to have a reasonable familiarity with the VVES programs and the HOOD toolkits core components.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus, e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title)

diesel engine vibration
vibration isolation
resilient suspension
visco elasticity

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM